FANUC Series 0*i*-MODEL F Plus

CONNECTION MANUAL (FUNCTION)

B-64693EN-1/01

The products in this manual are controlled based on Japan's "Foreign Exchange and Foreign Trade Law". The export from Japan may be subject to an export license by the government of Japan. Further, re-export to another country may be subject to the license of the government of the country from where the product is re-exported. Furthermore, the product may also be controlled by re-export regulations of the United States government.
Should you wish to export or re-export these products, please contact FANUC for advice.

The products in this manual are manufactured under strict quality control. However, when using any of the products in a facility in which a serious accident or loss is predicted due to a failure of the product, install a safety device.

In this manual we have tried as much as possible to describe all the various matters.
However, we cannot describe all the matters which must not be done, or which cannot be done, because there are so many possibilities.
Therefore, matters which are not especially described as possible in this manual should be regarded as "impossible".

SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units.
It is essential that these precautions be observed by users to ensure the safe operation of machines equipped with a CNC unit (all descriptions in this section assume this configuration). Note that some precautions are related only to specific functions, and thus may not be applicable to certain CNC units.
Users must also observe the safety precautions related to the machine, as described in the relevant manual supplied by the machine tool builder. Before attempting to operate the machine or create a program to control the operation of the machine, the operator must become fully familiar with the contents of this manual and relevant manual supplied by the machine tool builder.

DEFINITION OF WARNING, CAUTION, AND NOTE

This manual includes safety precautions for protecting the user and preventing damage to the machine. Precautions are classified into **Warning** and **Caution** according to their bearing on safety. Also, supplementary information is described as a **Note**. Read the **Warning**, **Caution**, and **Note** thoroughly before attempting to use the machine.

> ⚠ **WARNING**
> Used if a danger resulting in the death or serious injury of the user is expected to occur if he or she fails to observe the approved procedure.

> ⚠ **CAUTION**
> Used if a danger resulting in the minor or moderate injury of the user or equipment damage is expected to occur if he or she fails to observe the approved procedure.

> **NOTE**
> Used if a supplementary explanation not related to any of WARNING, and CAUTION is to be indicated.

- Read this manual carefully, and store it in a safe place.

GENERAL WARNINGS AND CAUTIONS

⚠ **WARNING**

1 Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

2 Before operating the machine, thoroughly check the entered data.
Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

3 Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate.
The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate.
If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

4 When using a tool compensation function, thoroughly check the direction and amount of compensation.
Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

5 The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change.
Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

⚠ CAUTION

1 Immediately after switching on the power, do not touch any of the keys on the MDI unit until the position display or alarm screen appears on the CNC unit. Some of the keys on the MDI unit are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.
2 The OPERATOR'S MANUAL and programming manual supplied with a CNC unit provide an overall description of the machine's functions. Note that the functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.
3 Some functions may have been implemented at the request of the machine-tool builder. When using such functions, refer to the manual supplied by the machine-tool builder for details of their use and any related cautions.
4 The liquid-crystal display is manufactured with very precise fabrication technology. Some pixels may not be turned on or may remain on. This phenomenon is a common attribute of LCDs and is not a defect.

NOTE

Programs, parameters, and macro variables are stored in non-volatile memory in the CNC unit. Usually, they are retained even if the power is turned off.
Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from non-volatile memory as part of error recovery.
To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.
The number of times to write machining programs to the non-volatile memory is limited.
You must use "High-speed program management" when registration and the deletion of the machining programs are frequently repeated in such case that the machining programs are automatically downloaded from a personal computer at each machining.
In "High-speed program management", the program is not saved to the non-volatile memory at registration, modification, or deletion of programs.

WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

This section covers the major safety precautions related to programming. Before attempting to perform programming, read the supplied OPERATOR'S MANUAL carefully such that you are fully familiar with their contents.

⚠ **WARNING**

1 **Coordinate system setting**
If a coordinate system is established incorrectly, the machine may behave unexpectedly as a result of the program issuing an otherwise valid move command. Such an unexpected operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.

2 **Positioning by nonlinear interpolation**
When performing positioning by nonlinear interpolation (positioning by nonlinear movement between the start and end points), the tool path must be carefully confirmed before performing programming. Positioning involves rapid traverse. If the tool collides with the workpiece, it may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3 **Function involving a rotation axis**
When programming normal-direction (perpendicular) control, pay careful attention to the speed of the rotation axis. Incorrect programming may result in the rotation axis speed becoming excessively high, such that centrifugal force causes the chuck to lose its grip on the workpiece if the latter is not mounted securely. Such mishap is likely to damage the tool, the machine itself, the workpiece, or cause injury to the user.

4 **Inch/metric conversion**
Switching between inch and metric inputs does not convert the measurement units of data such as the workpiece origin offset, parameter, and current position. Before starting the machine, therefore, determine which measurement units are being used. Attempting to perform an operation with invalid data specified may damage the tool, the machine itself, the workpiece, or cause injury to the user.

5 **Constant surface speed control**
When an axis subject to constant surface speed control approaches the origin of the workpiece coordinate system, the spindle speed may become excessively high. Therefore, it is necessary to specify a maximum allowable speed. Specifying the maximum allowable speed incorrectly may damage the tool, the machine itself, the workpiece, or cause injury to the user.

6 **Stroke check**
After switching on the power, perform a manual reference position return as required. Stroke check is not possible before manual reference position return is performed. Note that when stroke check is disabled, an alarm is not issued even if a stroke limit is exceeded, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

⚠ **WARNING**

7 **Tool post interference check**
A tool post interference check is performed based on the tool data specified during automatic operation. If the tool specification does not match the tool actually being used, the interference check cannot be made correctly, possibly damaging the tool or the machine itself, or causing injury to the user. After switching on the power, or after selecting a tool post manually, always start automatic operation and specify the tool number of the tool to be used.

8 **Same address command in same block**
The G code or M code including the same address cannot be commanded on the same block. If you use the same address, it may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user. Command on separate block.（About address P, refer to the appendix “List of functions include address P in the program command”）

⚠ **CAUTION**

1 **Absolute/incremental mode**
If a program created with absolute values is run in incremental mode, or vice versa, the machine may behave unexpectedly.

2 **Plane selection**
If an incorrect plane is specified for circular interpolation, helical interpolation, or a canned cycle, the machine may behave unexpectedly. Refer to the descriptions of the respective functions for details.

3 **Torque limit skip**
Before attempting a torque limit skip, apply the torque limit. If a torque limit skip is specified without the torque limit actually being applied, a move command will be executed without performing a skip.

4 **Programmable mirror image**
Note that programmed operations vary considerably when a programmable mirror image is enabled.

5 **Compensation function**
If a command based on the machine coordinate system or a reference position return command is issued in compensation function mode, compensation is temporarily canceled, resulting in the unexpected behavior of the machine. Before issuing any of the above commands, therefore, always cancel compensation function mode.

WARNINGS AND CAUTIONS RELATED TO HANDLING

This section presents safety precautions related to the handling of machine tools. Before attempting to operate your machine, read the supplied OPERATOR'S MANUAL carefully, such that you are fully familiar with their contents.

⚠ **WARNING**

1 **Manual operation**
When operating the machine manually, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and feedrate have been specified correctly. Incorrect operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

2 **Manual reference position return**
After switching on the power, perform manual reference position return as required.
If the machine is operated without first performing manual reference position return, it may behave unexpectedly. Stroke check is not possible before manual reference position return is performed.
An unexpected operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3 **Manual handle feed**
In manual handle feed, rotating the handle with a large scale factor, such as 100, applied causes the tool and table to move rapidly. Careless handling may damage the tool and/or machine, or cause injury to the user.

4 **Disabled override**
If override is disabled (according to the specification in a macro variable) during threading, rigid tapping, or other tapping, the speed cannot be predicted, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

5 **Origin/preset operation**
Basically, never attempt an origin/preset operation when the machine is operating under the control of a program. Otherwise, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the tool, or causing injury to the user.

6 **Workpiece coordinate system shift**
Manual intervention, machine lock, or mirror imaging may shift the workpiece coordinate system. Before attempting to operate the machine under the control of a program, confirm the coordinate system carefully.
If the machine is operated under the control of a program without making allowances for any shift in the workpiece coordinate system, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

⚠ **WARNING**

7 **Software operator's panel and menu switches**
Using the software operator's panel and menu switches, in combination with the MDI unit, it is possible to specify operations not supported by the machine operator's panel, such as mode change, override value change, and jog feed commands.
Note, however, that if the MDI unit keys are operated inadvertently, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

8 **RESET key**
Pressing the RESET key stops the currently running program. As a result, the servo axes are stopped. However, the RESET key may fail to function for reasons such as an MDI unit problem. So, when the motors must be stopped, use the emergency stop button instead of the RESET key to ensure security.

⚠ **CAUTION**

1 **Manual intervention**
If manual intervention is performed during programmed operation of the machine, the tool path may vary when the machine is restarted. Before restarting the machine after manual intervention, therefore, confirm the settings of the manual absolute switches, parameters, and absolute/incremental command mode.

2 **Feed hold, override, and single block**
The feed hold, feedrate override, and single block functions can be disabled using custom macro system variable #3004. Be careful when operating the machine in this case.

3 **Dry run**
Usually, a dry run is used to confirm the operation of the machine. During a dry run, the machine operates at dry run speed, which differs from the corresponding programmed feedrate. Note that the dry run speed may sometimes be higher than the programmed feed rate.

4 **Program editing**
If the machine is stopped, after which the machining program is edited (modification, insertion, or deletion), the machine may behave unexpectedly if machining is resumed under the control of that program. Basically, do not modify, insert, or delete commands from a machining program while it is in use.

WARNINGS RELATED TO PARAMETERS

⚠ **WARNING**

1 When machining a workpiece for the first time after modifying a parameter, close the machine cover. Never use the automatic operation function immediately after such a modification. Instead, confirm normal machine operation by using functions such as the single block function, feedrate override function, and machine lock function, or by operating the machine without mounting a tool and workpiece. If the machine is used before confirming that it operates normally, the machine may move unpredictably, possibly damaging the machine or workpiece, and presenting a risk of injury.

2 The CNC and PMC parameters are set to their optimal values, so that those parameters usually need not be modified. When a parameter must be modified for some reason, ensure that you fully understand the function of that parameter before attempting to modify it. If a parameter is set incorrectly, the machine may move unpredictably, possibly damaging the machine or workpiece, and presenting a risk of injury.

WARNINGS RELATED TO EDIT SCREENS FOR TOUCH PANEL

⚠ **WARNING**

FANUC’s touch panel is an analog resistive film type. When two or more points are pressed at the same time, there is a possibility that it behaves as if the center of these points was pressed, and this wrong output or malfunction may cause an accident. Do not create a virtual machine operator’s panel screens on which two or more points are pressed at the same time for touch panel operation.

On the virtual machine operator panel screen, never support safety-related operations that may lose human life or may cause serious damage, or real-time operations such as emergency stop, program start, program stop, axis movements, etc. If there is a failure in the CNC, peripheral units, or cable, wrong outputs or malfunctions may cause an accident. In addition, real-time operation is not guaranteed on the touch panel screen.

GENERAL WARNINGS FOR CNC APPLICATION DEVELOPMENT

⚠ WARNING

Be careful enough for the following warnings when you develop two or more applications or use networks.
If you neglect them, machine may behave in an unexpected working and tool, work piece, and machine may be damaged. In the worst case, there is an operator's extreme risk of death or serious injury.

1 Be careful enough if you write an identical NC data, an identical PMC data or a series of related data set by two or more above applications including network functions. Because they are executed based on each individual cycles (in other words, asynchronous cycles), there is a possibility that the data will be written in an unexpected order.

   Therefore, do NOT write above data in the following cases.
   - Applications and network functions
   - Two or more applications
   - Two or more network functions

   Data, applications and network functions of interest are listed in below. However, all may not be listed completely because new features will be added in the future.

2 Be careful enough that you must prevent PMC signals in the same byte from being written by the following two or more applications including network functions. While an application reads and writes one byte of PMC signals, other applications may write the same byte.

3 Be careful enough if you process a PMC signal set that is related to a NC function by using the following two or more applications including network functions. Because they are executed based on each individual cycles (in other words, asynchronous cycles), there is a possibility that the NC may receive the PMC signal set in an unexpected order.

4 Generally, when multi-byte data are read or written at once among the following two or more applications including network functions, the coherency of the read multi-byte data (in other words, reading all latest data at once) is not guaranteed. To ensure the coherency of the multi-byte data, prepare flags to notify the completion of reading or writing process that is separated from the entity of the data and make the handshaking process to access the data by using the flags.

⚠ **WARNING**

Data List Table

Category	Data
General data for NC	Parameter, Tool compensation value and related data, Work zero offset value and related data, Workpiece coordinate system shift value and related data, Macro variable, P-CODE variable, Program and related data, Tool management function data, Tool life management data, Error compensation related data , Overtravel check (Interference check) related data , Software operator's panel related data
PMC data	PMC signal, PMC parameter
Data for Laser, Punch press or Wire cut	Tool data for punch press and related data, Safety zone data and related data, Laser cutting condition data and related data, Laser oscillator setting data and related data, Wire consumption compensation data, Guide position compensation data, Workpiece leveling data
Other data	Parameters for Data Server, Parameters for network setting

List Table of Applications and Network Functions

Category	Functions
Applications	PMC Ladder, Macro Executor, C Language Executor, FANUC PICTURE, FOCAS2
Network functions	FL-net, EtherNet/IP, PROFINET, Modbus/TCP, PROFIBUS-DP, DeviceNet, CC-Link

5 CNC has functions that read or write PMC signals in other than the G/F address. Be careful enough if the above mentioned applications and network read or write PMC signals used by these functions. When reading or writing the same PMC signal, applications or CNC functions may work in an unexpected manner.
As for the CNC functions of interest, refer to the connection manual (Function) (B-64693EN-1) "Appendix B. List of Functions Using PMC Signals Other Than G/F Address".

WARNINGS RELATED TO DAILY MAINTENANCE

⚠ **WARNING**

1 **Memory backup battery replacement**
When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.
When opening the cabinet and replacing the batteries, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover). Touching the uncovered high-voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The CNC uses batteries to preserve the contents of its memory, because it must retain data such as programs, offsets, and parameters even while external power is not applied.
If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.
When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the CNC's memory will be lost.
Refer to the Section “Method of replacing battery” in the OPERATOR’S MANUAL (Common to T/M series) for details of the battery replacement procedure.

⚠ **WARNING**

2 **Absolute pulse coder battery replacement**

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.
When opening the cabinet and replacing the batteries, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching the uncovered high-voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The absolute pulse coder uses batteries to preserve its absolute position.
If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.
When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost.
Refer to the FANUC SERVO MOTOR αi series Maintenance Manual for details of the battery replacement procedure.

⚠ **WARNING**

3 **Fuse replacement**

Before replacing a blown fuse, however, it is necessary to locate and remove the cause of the blown fuse.
For this reason, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing a fuse with the cabinet open, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching an uncovered high-voltage circuit presents an extremely dangerous electric shock hazard.

⚠ **WARNING**

4 When using the controller unit, display unit, MDI unit, or machine operator's panel, prevent these units from directly exposing to chips or coolants. Even if direct exposure to coolants is prevented, coolants containing sulfur or chlorine at a high activation level, oil-free synthetic-type coolants, or water-soluble coolants at a high alkali level particularly have large effects on the control unit and peripheral units, possibly causing the following failures.

- Coolants containing sulfur or chlorine at a high activation level
  Some coolants containing sulfur or chlorine are at an extremely high activity level. If such a coolant adheres to the CNC or peripheral units, it reacts chemically with a material, such as resin, of equipment, possibly leading to corrosion or deterioration. If it gets in the CNC or peripheral units, it corrodes metals, such as copper and silver, used as component materials, possibly leading to a defective component.
- Synthetic-type coolants having a high permeability
  Some synthetic-type coolants whose lubricating component is, for example, PAG (polyalkylene glycol) have an extremely high permeability. If such a coolant is used even in equipment having a high closeness, it can readily flow into the CNC or peripheral units through, for example, gaskets. It is likely that, if the coolant gets in the CNC or a peripheral unit, it may deteriorate the insulation and damage the components.
- Water-soluble coolants at a high alkali level
  Some coolants whose pH is increased using alkanolamine are so strong alkali that its standard dilution will lead to pH10 or higher. If such a coolant spatters over the surface of the CNC or peripheral unit, it reacts chemically with a material, such as resin, possibly leading to corrosion or deterioration.

PREFACE

Organization of this manual

This manual describes all the NC functions required to enable machine tool builders to design their CNC machine tools. The following items are explained for each function.

1. Overview

Describes feature of the function. Refer to Operator’s Manual as required.

2. Signal

Describes names, functions, output conditions and addresses of the signals required to realize a function.

3. Parameter

Describes parameters related with a function.

4. Alarms and message

Lists the alarms and messages related with a function in a table.

5. Reference item

List the related items of the related manuals in a table.

A list of addresses of all signals and a list of signals are described in the appendix of this manual. Refer to it as required.

Applicable models

The models covered by this manual, and their abbreviations are :

Model name	Abbreviation		
FANUC Series 0*i*-TF Plus	0*i*-TF Plus	Series 0*i*-F Plus	Series 0*i*
FANUC Series 0*i*-MF Plus	0*i*-MF Plus		

> **NOTE**
> 1 For an explanatory purpose, the following descriptions may be used according to the CNC model :
> - 0*i*-TF Plus : Lathe system (T series)
> - 0*i*-MF Plus : Machining center system (M series)
>
> 2 Some functions described in this manual may not be applied to some products. For details, refer to the DESCRIPTIONS (B-64692EN). ".

Description of symbols

The following symbols are used in this manual. These symbols are described below.

- M

Indicates a description that is valid only for the machine center system (M series).
In a general description of the method of machining, a machining center system operation is identified by a phase such as "for milling machining".

- T

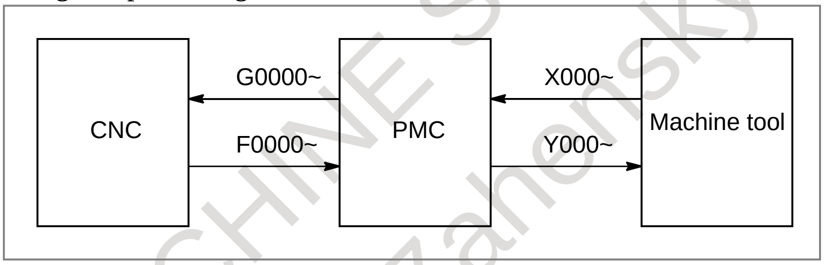
Indicates a description that is valid only for the lathe system (T series).
In a general description of the method of machining, a lathe system operation is identified by a phrase such as "for lathe cutting".

-

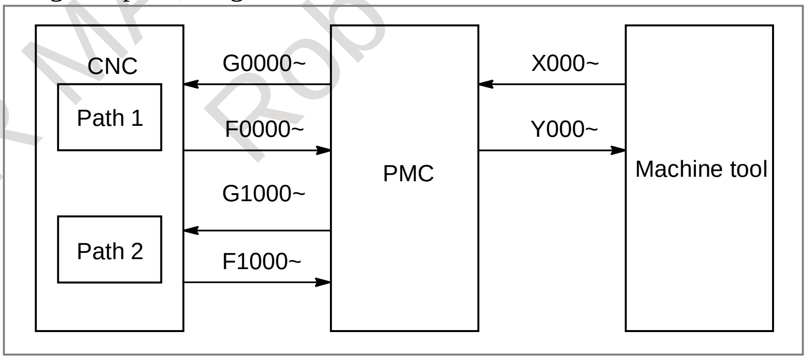
Indicates the end of a description of a system control type.
When a system control type mark mentioned above is not followed by this mark, the description of the system control type is assumed to continue until the next item or paragraph begins. In this case, the next item or paragraph provides a description common to the control types.

Description of signals

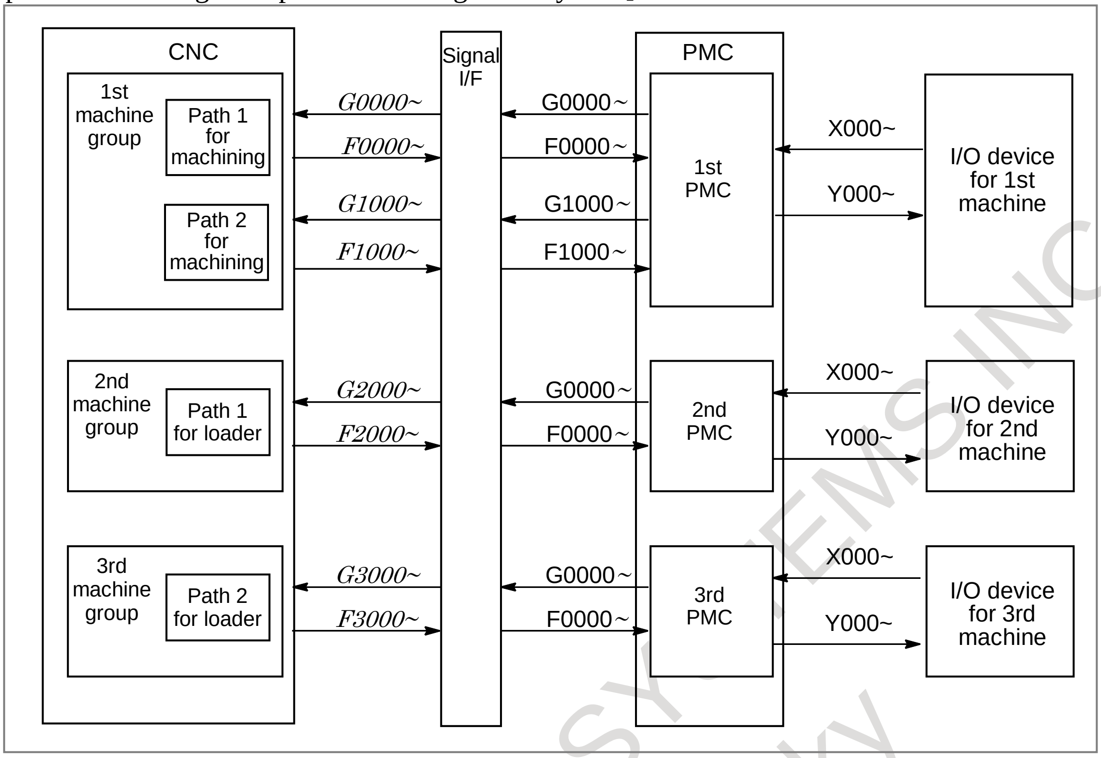
[Example of controlling one path using one PMC]



[Example of controlling two path using one PMC]



[Example of controlling multipath CNC using PMC system]



> **NOTE**
> Each PMC of a multipath PMC system has an independent signal area. The F, G, X, and Y signal addresses of each PMC begin with 0. On the other hand, the F and G signal addresses from the viewpoint of the CNC are fixed for each path number. Note that the F and G signal addresses used in programming of each ladder are different from those from the viewpoint of the CNC.

- Expression of signals

Address / Symbol (#0 to #7 indicates bit position)

Address	#7	#6	#5	#4	#3	#2	#1	#0
Fn000	OP	SA	STL	SPL				RWD

In an item where both lathe system (T series) and machining center system (M series) are described, some signals are covered with shade ( ) in the signal address figure as shown below. This means either lathe system or machining center system does not have this signal. Upper part is for lathe system and lower part is for machining center system.

	#7	#6	#5	#4	#3	#2	#1	#0	
Gn053	*CDZ		ROVLP		UINT			TMRON	T series M series

[Example 1]

The figure above indicates *CDZ is provided only for the lathe system while the other signals for both the lathe system (T series) and machining system (M series).

	#7	#6	#5	#4	#3	#2	#1	#0	
Gn040					OFN9	OFN8	OFN7	OFN6	T series M series

[Example 2]

Signals OFN6 to OFN9 are for machining center system (M series) only.

NOTE

1 The following notational conventions are used in the signal description of each function.

Example) Axis moving signals MV1 to MV8 <Fn102>

↑ ↑ ↑

Signal name Symbol name Signal address

2 For multipath control, one of the following superscripts is attached to the top right of a symbol depending on the signal type.
- Path type (for path 1 on PMC side) : #1
- Path type (for path 2 on PMC side) : #2
- Path type (for path 3 on PMC side) : #3
- Path type : #P
- Controlled axis type : #SV
- Spindle type : #SP
- PMC axis control group type : #PX

Refer to Appendix “List of Addresses“ for details

3 For the signals, a single data number is assigned to 8 bits. Each bit has a different meaning.

4 The letter "n" in each address representation indicates the address position used in each path on the CNC side, as shown below.

1st path : n=0 (No. 0 to 999)
2nd path : n=1 (No. 1000 to 1999)
3rd path : n=2 (No. 2000 to 2999)

5 For a signal of controlled axis type, when the number of axes exceeds eight for each path, set parameter No.3021 to address this situation.

Description of parameters

Parameters are classified by data type as follows:

Data type	Valid data range	Remarks
Bit Bit machine group Bit path Bit axis Bit spindle	0 or 1	
Byte Byte machine group Byte path Byte axis Byte spindle	-128 to 127 0 to 255	Some parameters handle these types of data as unsigned data.
Word Word machine group Word path Word axis Word spindle	-32768 to 32767 0 to 65535	Some parameters handle these types of data as unsigned data.
2-word 2-word machine group 2-word path 2-word axis 2-word spindle	0 to ±999999999	Some parameters handle these types of data as unsigned data.
Real Real machine group Real path Real axis Real spindle	See the Standard Parameter Setting Tables.	

NOTE

1. Each of the parameters of the bit, bit machine group, bit path, bit axis, and bit spindle types consists of 8 bits for one data number (parameters with eight different meanings).
2. For machine group types, parameters corresponding to the maximum number of machine groups are present, so that independent data can be set for each machine group.
3. For path types, parameters corresponding to the maximum number of paths are present, so that independent data can be set for each path.
4. For axis types, parameters corresponding to the maximum number of control axes are present, so that independent data can be set for each control axis.
5. For spindle types, parameters corresponding to the maximum number of spindles are present, so that independent data can be set for each spindle axis.
6. The valid data range for each data type indicates a general range. The range varies according to the parameters. For the valid data range of a specific parameter, see the explanation of the parameter.

- Standard parameter setting tables

This section defines the standard minimum data units and valid data ranges of the CNC parameters of the real type, real machine group type, real path type, real axis type, and real spindle type. The data type and unit of data of each parameter conform to the specifications of each function.

(A) Length and angle parameters (type 1)

Unit of data	Increment system	Minimum data unit	Valid data range
mm deg.	IS-A	0.01	-999999.99 to +999999.99
	IS-B	0.001	-999999.999 to +999999.999
	IS-C	0.0001	-99999.9999 to +99999.9999
inch	IS-A	0.001	-99999.999 to +99999.999
	IS-B	0.0001	-99999.9999 to +99999.9999
	IS-C	0.00001	-9999.99999 to +9999.99999

(B) Length and angle parameters (type 2)

Unit of data	Increment system	Minimum data unit	Valid data range
mm deg.	IS-A	0.01	0.00 to +999999.99
	IS-B	0.001	0.000 to +999999.999
	IS-C	0.0001	0.0000 to +99999.9999
inch	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
	IS-C	0.00001	0.00000 to +9999.99999

(C) Velocity and angular velocity parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min deg/min	IS-A	0.01	0.0 to +999000.00
	IS-B	0.001	0.0 to +999000.000
	IS-C	0.0001	0.0 to +99999.9999
inch/min	IS-A	0.001	0.0 to +96000.000
	IS-B	0.0001	0.0 to +9600.0000
	IS-C	0.00001	0.0 to +4000.00000

If bit 7 (IESP) of parameter No. 1013 is set to 1, the valid data ranges for IS-C are extended as follows:

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min deg/min	IS-C	0.001	0.000 to +999000.000
inch/min	IS-C	0.0001	0.0000 to +9600.0000

(D) Acceleration and angular acceleration parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/sec^2 deg./sec^2	IS-A	0.01	0.00 to +999999.99
	IS-B	0.001	0.000 to +999999.999
	IS-C	0.0001	0.0000 to +99999.9999
inch/sec^2	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
	IS-C	0.00001	0.00000 to +9999.99999

If bit 7 (IESP) of parameter No. 1013 is set to 1, the valid data ranges for IS-C are extended as follows:

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min deg/min	IS-C	0.001	0.000 to +999999.999
inch/min	IS-C	0.0001	0.0000 to +99999.9999

⚠ CAUTION

1. Values are rounded up or down to the nearest multiples of the minimum data unit.
2. A valid data range means data input limits, and may differ from values representing actual performance.
3. For information on the ranges of commands to the CNC, refer to Appendix D, "Range of Command Value" of the Operator’s Manual (B-64694EN).

- Parameters of the bit type, bit machine group type, bit path type, bit axis type, and bit spindle type

Data No.	#7	#6	#5	#4	#3	#2	#1	#0
0000			SEQ			INI	ISO	TVC

Data (Data #0 to #7 are bit positions.)

- Parameters other than the bit-type parameters above

Data No.	Data
1023	Number of the servo axis for each axis

NOTE

1 The bits left blank in “description of parameters” and parameter numbers that appear on the display but are not found in the parameter list are reserved for future expansion. They must always be 0.

2 A parameter usable with only one path control type, namely, the lathe system (T series) or the machining center system (M series), is indicated using two rows as shown below. When a row is blank, the parameter is not usable with the corresponding series.

[Example 1]

Parameter HTG is a parameter common to the lathe system (T series) or the machining center system (M series), but Parameters RTV and ROC are parameters valid only for the lathe system (T series).

	#7	#6	#5	#4	#3	#2	#1	#0	
1403	RTV		HTG	ROC					T series
			HTG						M series

[Example 2]

The following parameter is provided only for the machining center system (M series).

1411		T series
	Cutting feedrate	M series

3 When "to" is inserted between two parameter numbers, there are parameters with successive numbers between the two starting and ending parameter numbers, but those intermediate parameter numbers are omitted for convenience.

4 The lower-case letter "x" or "s" following the name of a bit-type parameter indicates the following:

- ”□□□x” : Bit axis type parameters
- ”○○○s” : Bit spindle type parameters

Related manuals of Series 0*i*-F plus

The following table lists the manuals related to Series 0*i*-F Plus. This manual is indicated by an asterisk(*).

Table 1 Related manuals of Series 0*i*-F Plus

Manual name	Specification number	
DESCRIPTIONS	B-64692EN	
CONNECTION MANUAL (HARDWARE)	B-64693EN	
CONNECTION MANUAL (FUNCTION)	B-64693EN-1	*
OPERATOR'S MANUAL (Common to Lathe System/Machining Center System)	B-64694EN	
OPERATOR'S MANUAL (For Lathe System)	B-64694EN-1	
OPERATOR'S MANUAL (For Machining Center System)	B-64694EN-2	
MAINTENANCE MANUAL	B-64695EN	
PARAMETER MANUAL	B-64700EN	
Programming		
Macro Executor PROGRAMMING MANUAL	B-63943EN-2	
Macro Compiler PROGRAMMING MANUAL	B-66263EN	
C Language Executor PROGRAMMING MANUAL	B-63943EN-3	
PMC		
PMC PROGRAMMING MANUAL	B-64513EN	
Network		
PROFIBUS-DP Board CONNECTION MANUAL	B-63993EN	
Industrial Ethernet CONNECTION MANUAL	B-64013EN	
Fast Ethernet / Fast Data Server OPERATOR'S MANUAL	B-64014EN	
DeviceNet Board CONNECTION MANUAL	B-64043EN	
FL-net Board CONNECTION MANUAL	B-64163EN	
CC-Link Board CONNECTION MANUAL	B-64463EN	
Operation guidance function		
MANUAL GUIDE *i* (Common to Lathe System/Machining Center System) OPERATOR'S MANUAL	B-63874EN	
MANUAL GUIDE *i* (For Machining Center System) OPERATOR'S MANUAL	B-63874EN-2	
MANUAL GUIDE *i* (Set-up Guidance Functions) OPERATOR'S MANUAL	B-63874EN-1	
MANUAL GUIDE 0*i* OPERATOR'S MANUAL	B-64434EN	
Dual Check Safety		
Dual Check Safety CONNECTION MANUAL	B-64483EN-2	

Related manuals of SERVO MOTOR α*i*/β*i* series

The following table lists the manuals related to SERVO MOTOR α*i*/β*i* series

Table 2 Related manuals of SERVO MOTOR α*i*/β*i* series

Manual name	Specification number
FANUC AC SERVO MOTOR α*i*-B series FANUC AC SERVO MOTOR α*i* series DESCRIPTIONS	B-65262EN
FANUC AC SPINDLE MOTOR α*i*-B / β*i*-B series DESCRIPTIONS	B-65452EN
FANUC AC SERVO MOTOR β*i*-B series FANUC AC SERVO MOTOR β*i* series DESCRIPTIONS	B-65302EN
FANUC SERVO AMPLIFIER α*i*-B series DESCRIPTIONS	B-65412EN
FANUC SERVO AMPLIFIER β*i*-B series DESCRIPTIONS	B-65422EN
FANUC AC SERVO MOTOR α*i* series FANUC AC SPINDLE MOTOR α*i* series FANUC SERVO AMPLIFIER α*i* series MAINTENANCE MANUAL	B-65285EN
FANUC AC SERVO MOTOR β*i* series FANUC AC SPINDLE MOTOR β*i* series FANUC SERVO AMPLIFIER β*i* series MAINTENANCE MANUAL	B-65325EN
FANUC AC SERVO MOTOR α*i* series FANUC AC SERVO MOTOR β*i* series FANUC LINEAR MOTOR L*i*S series FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR D*i*S series PARAMETER MANUAL	B-65270EN
FANUC AC SPINDLE MOTOR α*i*/β*i* series, BUILT-IN SPINDLE MOTOR B*i* series PARAMETER MANUAL	B-65280EN

The above servo motors and the corresponding spindles can be connected to the CNC covered in this manual.
This manual mainly assumes that the FANUC SERVO MOTOR α*i* series of servo motor is used. For servo motor and spindle information, refer to the manuals for the servo motor and spindle that are actually connected.

Notes on various kinds of data

NOTE

Machining programs, parameters, macro variables, etc. are stored in the CNC unit internal non-volatile memory. In general, these contents are not lost by the switching ON/OFF of the power. However, it is possible that a state can occur where precious data stored in the non-volatile memory has to be deleted, because of inadvertent erasing, or by a failure restoration. In order to restore rapidly when this kind of mishap occurs, it is recommended that you create a copy of the various kinds of data beforehand.
The number of times to write machining programs to the non-volatile memory is limited.
You must use "High-speed program management" when registration and the deletion of the machining programs are frequently repeated in such case that the machining programs are automatically downloaded from a host computer at each machining.
In "High-speed program management", the program is not saved to the non-volatile memory at registration, modification, or deletion of programs.
Please make the application software by using FOCAS2/ C Language Library to save the changed programs to the non-volatile memory when "High-speed program management" is used.

TABLE OF CONTENTS

APPENDIX

1 AXIS CONTROL

1.1 CONTROLLED AXIS

Overview

The axis composition of Series 0*i*-F Plus is as shown in Table1.1 (a). "Total number of control axes" is sum of "Number of feed axes" and "Number of spindle axes".

Table1.1 (a) Axis composition of Series 0*i*-F Plus

Item		Series 0*i*-MF Plus			Series 0*i*-TF Plus		
		Type 0 Type 1 1-path system	Type 0 Type 1 2-path system	Type 3 Type 5	Type 0 Type 1 1-path system	Type 0 Type 1 2-path system	Type 3 Type 5
Maximum total number of control axes (total/each path)		9 axes	11 axes/ 9 axes	6 axes	9 axes	11 axes/ 9 axes	6 axes
Maximum number of feed axes *1 (total/each path)	Basic	5 axes	8 axes/ 5 axes	5 axes	4 axes	8 axes/5 axes	4 axes
	Controllable axes expansion	7 axes	9 axes/ 7 axes	-	7 axes	9 axes/7 axes	5 axes
Maximum number of spindle axes (total/each path)	Basic	2 axes	2 axes/ 2 axes	1 axis	2 axes	2 axes/2 axes	1 axis
	Spindle axes expansion	-	4 axes/ 3 axes	-	3 axes	4 axes/3 axes	2 axes
Simultaneously controlled axes (each path)		4 axes in the time	4 axes in the time	4 axes in the time	4 axes in the time	4 axes in the time	4 axes in the time
PMC axis control *2 (total)		4 axes int the time	8 axes in the time	4 axes int the time	4 axes int the time	8 axes in the time	4 axes int the time
Number of connected servo motors*3 (HRV3 control/HRV2 control)		10/12	10/12	5	10/12	10/12	5

*1: With PMC Axis. Without "Cs Contour control Axis" and "Spindle control axis with servo motor".
*2: Without "Cs Contour control Axis"
*3: Different according to the main-board.
Total of all path including loader path.
The following are included.
- Servo axis(Including PMC Axis)
- Spindle Control axis with Servo Motor
- EGB dummy axis
- Serial feedback dummy axis

> **NOTE**
> 1. "Controllable axes expansion" and "Spindle axes expansion" are optional function.
> 2. The number of connected servo motors is different according to the main-board.

Whether it is counted by feed-axes/spindle-axes is decided depending on the kind of each axis/spindle. How to count axis/spindle is as shown in Table1.1 (b).

Table1.1 (b) How to count axis/spindle

Item		Number of Feed Axes	Number of Spindle Axes	Total Number of Control Axes (Number of Feed Axes + Number of Spindle Axes)	Number of Connected Servo Motors
Servo Axis(including PMC Axis)		1	0	1	1
Analog Spindle*1		0	1	1	0
Serial Spindle		0	1	1	0
Cs Contour Control	Cs Axis	0	1	1	0
	Virtual Cs Axis*2*3	0	0	0	0
Axis that is controlled by "Spindle Control with Servo Motor"		0	1	1	1
EGB Dummy Axis*3		0	0	0	1*4
Serial Feedback Dummy Axis*3		0	0	0	1*4

*1: Up to 1 axis in the total.
*2: Up to 2 axes in the total. Up to 1 axes in each path.
*3: The total of the following numbers of axes is up to 3 axes in the total.
- Virtual Cs axis
- EGB dummy axis
- Serial feedback dummy axis
*4: It is counted by the number of connected servo motors though the motor is not connected.

Loader Path

When the loader control function is used, the loader path can be used.
In the loader path, the servo axis can be used up to 3 axes. Other axis/spindle cannot be used.
The axis in the loader path is contained in the number of connected servo motors. Refer to Table1.1 (a) for the maximum number of connected servo motors.
The axis in the loader path is not contained in the "Total number of control axes" and "Number of feed axes".

> **NOTE**
> "Function for loader control" is optional function. The number of the usable loader path is different depending on the option composition.

Alarm and message

Number	Message	Description
PS0015	TOO MANY SIMULTANEOUS AXES	A move command was specified for more axes than can be controlled by simultaneous axis control. Either add on the simultaneous axis control extension option, or divide the number of programmed move axes into two blocks.
DS0050	TOO MANY SIMULTANEOUS AXES	A movement was performed along more axes than can be controlled by simultaneous axis control. Check whether a command in the program is specified for more axes than can be controlled by simultaneous axis control.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Number of controlled axes
CONNECTION MANUAL (FUNCTION) (This manual)	Multipath control

1.2 SETTING EACH AXIS

1.2.1 Name of Axes

Overview

Each axis that is controlled by the CNC (including those controlled by the PMC) must be named.
To name an axis, select a desired character from among A, B, C, U, V, W, X, Y, and Z and set the character as the first axis name character (parameter No. 1020).

> **NOTE**
> 1 The same axis name cannot be assigned to more than one axis. (The same axis name can be used on different paths.)
> 2 With the lathe system, when G code system A is used, neither U, V, nor W can be used as an axis name. Only when G code system B or C is used, U, V, and W can be used as axis names.
> 3 When a multiple repetitive canned cycle is used, only X, Y, or Z can be used as the address of a target axis.

- Extended axis name

The extended axis name function can be used to use an axis name consisting of up to three characters.
To use an extended axis name:
<1> Enables the extended axis name function (bit 0 (EEA) of parameter No. 1000 is set to 1).
<2> Set the first character (A, B, C, U, V, W, X, Y, or Z) in parameter No. 1020 (first axis name character).
<3> Set the second character ('0' to '9' and 'A' to 'Z') in parameter No. 1025 (second axis name character).
<4> Set the third character ('0' to '9' and 'A' to 'Z') in parameter No. 1026 (third axis name character).

> **NOTE**
> 1 If the second axis name character is not set for an axis, the third axis name character is invalid.
> 2 When setting 0 to 9 for the second axis name character, do not set A to Z for the third axis name character.
> 3 When an axis name ends with a numeric character, an equal sign (=) is required to be specified between the axis name and the command value.
> 4 In a macro call, no extended axis name can be used as an argument.
> 5 If at least one axis in a path uses an extended axis name when bit 2 (EAS) of parameter No.11308 is set to 0, subscripts (parameter No. 3131) cannot be used for axis names in the path.
> 6 When G code system A is used for a lathe system, X, Y, Z, or C may be used for the first axis name character of an axis. In this case, when a command containing U, V, W, or H as the first axis name character is specified, it is used as the incremental command for the corresponding axis.
> 7 In a multipath system, if an extended axis name is not used on a path or if bit 2 (EAS) of parameter No. No.11308 is set to 1 and subscripts (parameter No. 3131) are not set for axis names, the path number will automatically be the subscript for axis names. To disable the display of axis name subscripts, set a blank (32) of ASCII code in the parameter for specifying an axis name subscript.

> **NOTE**
> 8 If the custom macro function is enabled, the same extended axis name as a reserved word cannot be used. Such an extended axis name is regarded as a reserved word.
> Because of reserved words of custom macros, extended axis names that start with the following combinations of two characters cannot be used:
> AB, AC, AD, AN, AS, AT, AX, BC, BI, BP, CA, CL, CO, US, WH, WR, XO, ZD, ZE, ZO, ZW

	First axis name character (No. 1020)	Second axis name character (No. 1025)	Third axis name character (No. 1026)
Setting	A, B, C, U, V, W, X, Y, Z	0 to 9	0 to 9
		A to Z	0 to 9 A to Z
Correct example <1>	X	1	1
Correct example <2>	X	A	1
Correct example <3>	X	A	B
Incorrect example	X	1	A

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1000								**EEA**

[Input type] Parameter input
[Data type] Bit

#0 EEA An extended axis name and extended spindle name are:
0: Invalid
1: Valid

1020	**Program axis name for each axis**

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 65 to 67 ,85 to 90

An axis name (axis name 1: parameter No. 1020) can be arbitrarily selected from 'A', 'B', 'C', 'U', 'V', 'W', 'X', 'Y', and 'Z'. (When G code system A is used with the lathe system, however, 'U', 'V', and 'W' are not selectable.) When bit 0 (EEA) of parameter No. 1000 is set to 1, the length of an axis name can be extended to three characters by setting axis name 2 (parameter No. 1025) and axis name 3 (parameter No. 1026) (extended axis name).
For axis names 2 and 3, a character from '0' to '9' and 'A' to 'Z' of ASCII code can be arbitrarily selected. However, the setting of axis name 3 for each axis is invalid if axis name 2 is not set. Moreover, if a character from '0' to '9' is set as axis name 2, do not use a character from 'A' to 'Z' as axis name 3.

(Tip) ASCII code

Axis name	X	Y	Z	A	B	C	U	V	W
Setting	88	89	90	65	66	67	85	86	87

When G code system A is used with the lathe system, and the character 'X','Y','Z', or 'C' is used as axis name 1 of an axis, a command with 'U','V','W', or 'H' specified for axis name 1 represents an incremental command for the axis.

NOTE
1. When the setting value is out of range, it can not be recognized as an axis name.
2. When a multiple repetitive canned cycle is used, no character other than 'X','Y', and 'Z' can be used as the address of the target axis.
3. When the custom macro function is enabled, the same extended axis name as a reserved word cannot be used. Such an extended axis name is regarded as a reserved word.
   Because of reserved words of custom macros, extended axis names that start with the following two characters cannot be used:
   AB, AC, AD, AN, AS, AT, AX, BC, BI, BP, CA, CL, CO, US, WH, WR, XO, ZD, ZE, ZO, ZW
4. In a macro call, no extended axis name can be used as an argument.

1025	Program axis name 2 for each axis

1026	Program axis name 3 for each axis

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 48 to 57, 65 to 90

When axis name extension is enabled (when bit 0 (EEA) of parameter No. 1000 is set to 1), the length of an axis name can be extended to a maximum of three characters by setting axis name 2 and axis name 3. For axis names 2 and 3, a character from '0' to '9' and 'A' to 'Z' of ASCII code can be arbitrarily selected. However, the setting of axis name 3 for each axis is invalid if axis name 2 is not set. Moreover, if a character from '0' to '9' is set as axis name 2, do not use a character from 'A' to 'Z' as axis name 3.

3131	Subscript of axis name

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 9, 65 to 90

In order to distinguish axes under synchronization control and tandem control, specify a subscript for each axis name.

Setting value	Meaning
0	Each axis is set as an axis other than a synchronization control axis and tandem control axis.
1 to 9	A set value is used as a subscript.
65 to 90	A set letter (ASCII code) is used as a subscript.

[Example] When the axis name is X, a subscript is added as indicated below.

Setting value	Axis name displayed on a screen such as the position display screen
0	X
1	X1
77	XM
83	XS

If a multi-path system is used, no extended axis name is used within a path, or bit 2 (EAS) of parameter No. 11308 is set to 1 and no subscript is set for the axis names, then the path number is automatically used as the subscript for the axis names. To disable the display of axis name subscripts, set a blank (32) of ASCII code in the parameter for specifying an axis name subscript.

> **NOTE**
> If even one axis in a path uses an extended axis name when bit 2 (EAS) of parameter No. 11308 is set to 0, subscripts cannot be used for axis names in the path.

[Example]

- Example of setting an axis name
  When No. 1020 = 88, No. 1025 = 0, and No. 1026 = 0, the axis name is set to X.
  When No. 1020 = 88, No. 1025 = 65, and No. 1026 = 0, the axis name is set to XA.
  When No. 1020 = 88, No. 1025 = 66, and No. 1026 = 65, the axis name is set to XBA.
  When No. 1020 = 89, No. 1025 = 49, and No. 1026 = 0, the axis name is set to Y1.
  When No. 1020 = 90, No. 1025 = 49, and No. 1026 = 48, the axis name is set to Z10.
  When No. 1020 = 90, No. 1025 = 0, and No. 1026 = 65, the axis name is set to Z.

- Commands having a number at the end of the axis name
  Y1=100.
  Z10=200.

- Commands having an alphabet at the end of the axis name
  X100. or X=100.
  XA200. or XA=200.
  XBA300. or XBA=300.

- Incremental commands of lathe system G-code system A

Absolute command	Incremental command
XA100.	UA100.
Y1=200.	V1=200.
ZC300.	WC300.
C10=400.	H10=400.

- Relationship between the axis names and their settings

Axis name	Setting	Axis name	Setting	Axis name	Setting	Axis name	Setting
0	48	9	57	I	73	R	82
1	49	A	65	J	74	S	83
2	50	B	66	K	75	T	84
3	51	C	67	L	76	U	85
4	52	D	68	M	77	V	86
5	53	E	69	N	78	W	87
6	54	F	70	O	79	X	88
7	55	G	71	P	80	Y	89
8	56	H	72	Q	81	Z	90

Alarm and message

Number	Message	Description
PS0009	IMPROPER NC-ADDRESS	An illegal address was specified, or parameter 1020 is not set.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Axis name

1.2.2 Increment System

Overview

The increment system consists of the least input increment (for input unit) and least command increment (for output unit, machine unit). The least input increment is the least increment for programming the travel distance. The least command increment is the least increment for moving the tool on the machine. Both increments are represented in mm, inches, or degrees.
There are three types of increment systems as listed in Table 1.2.2 (a). A desired type can be set for each axis using the corresponding bit 0 (ISA), or 1 (ISC) of parameter No. 1013.
The least input increment can be set to metric input or inch input using the G code (G20 or G21) or setting parameter (bit 2 (INI) of parameter No. 0000).
The least command increment is set to either metric or inch system depending on the machine tool in advance. Select the metric or inch system using bit 0 (INM) of parameter No. 1001 in advance.
Any combined use of the inch and metric systems is not allowed. There are functions that cannot be used across axes with different increment systems (such as circular interpolation and cutter compensation).

NOTE
1 The unit (mm or inch) in the table is used for indicating a diameter value for diameter programming (when bit 3 (DIA) of parameter No. 1006 is set to 1) or a radius value for radius programming.
2 Even if the parameters bit 0 (ISA), or 1 (ISC) of parameter No.1013 is changed, the tool offset unit is not changed. Set the tool offset unit by parameters bit 0 (OFA), or 1 (OFC) of parameter No.5042.
3 Some increment systems are unavailable depending on the model. For details, refer to "DESCRIPTIONS" (B-64692EN).

Table 1.2.2 (a) Increment system

Name of an increment system	Least input increment		Least command increment		Maximum stroke	
IS-A	0.01	mm	0.01	mm	±999999.99	mm
	0.001	inch	0.001	inch	±99999.999	inch
	0.01	deg	0.01	deg	±999999.99	deg
IS-B	0.001	mm	0.001	mm	±999999.999	mm
	0.0001	inch	0.0001	inch	±99999.9999	inch
	0.001	deg	0.001	deg	±999999.999	deg
IS-C	0.0001	mm	0.0001	mm	±99999.9999	mm
	0.00001	inch	0.00001	inch	±9999.99999	inch
	0.0001	deg	0.0001	deg	±99999.9999	deg

When bit 7 (IPR) of parameter No. 1004, which multiplies the input increment by 10, is set to 1 and a value is specified with no decimal point, the specifications of each increment system are changed as listed in Table1.2.2 (b).

Table1.2.2 (b)

Name of an increment system	Least input increment		Least command increment		Maximum stroke	
IS-B	0.01	mm	0.001	mm	±999999.999	mm
	0.001	inch	0.0001	inch	±99999.9999	inch
	0.01	deg	0.001	deg	±999999.999	deg

Name of an increment system	Least input increment	Least command increment	Maximum stroke
IS-C	0.001 mm	0.0001 mm	±99999.9999 mm
	0.0001 inch	0.00001 inch	±9999.99999 inch
	0.001 deg	0.0001 deg	±99999.9999 deg

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0000						INI		

[Input type] Setting input
[Data type] Bit path

#2 INI Unit of input
0: In metrics
1: In inches

	#7	#6	#5	#4	#3	#2	#1	#0
1001								INM

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 INM Least command increment on the linear axis
0: In mm (metric system machine)
1: In inches (inch system machine)

	#7	#6	#5	#4	#3	#2	#1	#0
1004	IPR							

[Input type] Parameter input
[Data type] Bit path

#7 IPR When a number with no decimal point is specified, the least input increment of each axis is:
0: Not 10 times greater than the least command increment
1: 10 times greater than the least command increment
When the increment system is IS-A, and bit 0 (DPI) of parameter No. 3401 is set to 1 (fixed-point format), the least input increment cannot be 10 times greater than the least command increment.

	#7	#6	#5	#4	#3	#2	#1	#0
1006					DIAx			

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#3 DIAx The move command for each axis is based on:

0: Radius specification
1: Diameter specification

	#7	#6	#5	#4	#3	#2	#1	#0
1013							ISCx	ISAx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 ISAx
#1 ISCx Increment system of each axis

Increment system	#1 ISC	#0 ISA
IS-A	0	1
IS-B	0	0
IS-C	1	0

Reference item

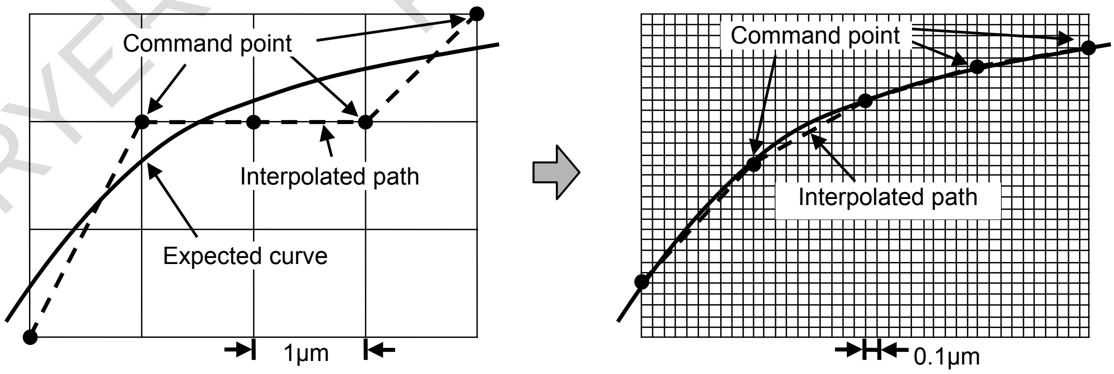
Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Increment system

1.2.2.1 High precision program command

Overview

In the program command to the axis address, the value that is one digit smaller than incremental system is enabled to command and analysis.

Features) Workpiece composed of free-form surface can be reproduced in high precision
- Incremental system is not changed, but program command unit is distributed to high precision
- The high precision machining can be achieved with keeping the usage of the machine
- In the machine of incremental system 1µm, the machining program 0.1µm is executable



Incremental system B (1µm)

Resolution of program command 0.1µm
(Incremental system C equally)
-> It becomes near expected curve.

Fig. 1.2.2.1 (a) Example in incremental system B

This function is enabled in all increment system (IS-A to C).
This function is activated when bit 0 (HPC) of parameter No.10478 is set to 1.

> **NOTE**
> 1 This function is a basic function.
> 2 This function is enabled to the axis address command (X,Y,Z, etc). It is invalid to the I, J, K, R command of circular interpolation etc.
> 3 This function doesn't relate to the detection unit.

Explanation

- Normal (When bit 0 (HPC) of parameter No.10478 is set to 0)

Conventionally, the axis address command of program has been rounded off a digit that was one digit smaller than the least incremental system.
When the command below least incremental system is described, it is became command of least incremental system.

Example) In case that input unit is metric, and increment system is 1μm (IS-B)
Program command

```
X10.0004    ->  X10.000
X10.0005    ->  X10.001
X10.00043   ->  X10.000
X10.00056   ->  X10.001
```

- High precision program command(When bit 0 (HPC) of parameter No.10478 is set to 1)

In the program command to the axis address with decimal point, the value that is one digit smaller than incremental system is enabled to command and analysis.
A digit that is 2 digits smaller than the least increment is system rounded off.
It is inputted the command in the unit to a digit that is 1 digit smaller than the least increment system.
This function is enabled for each program command of MEM / MDI / RMT mode.

Example) In case that input unit is metric, and increment system is 1μm (IS-B)
Program command

```
X10.0004    ->  X10.0004
X10.0005    ->  X10.0005
X10.00043   ->  X10.0004
X10.00056   ->  X10.0006
```

> **NOTE**
> When the bit 7 (IPR) of parameter No.1004 set to 1 (the least input increment is 10 times greater than the least command increment) and a number with no decimal point is specified, this function is disabled. The axis command value below incremental system is effective.

- Program example

Program command)

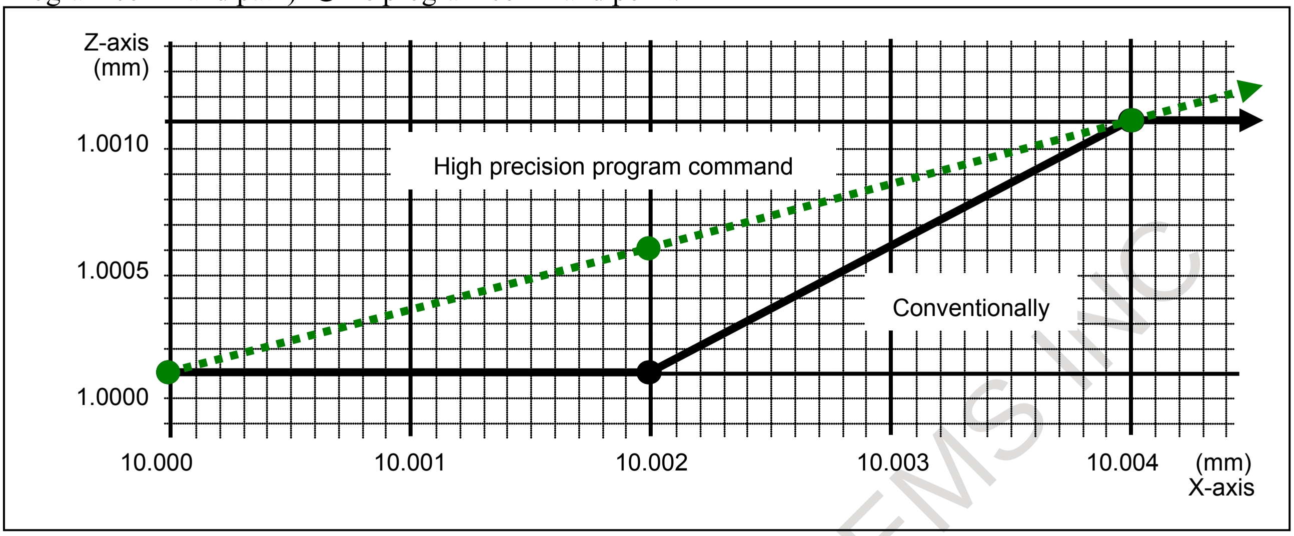
Conventionally command
(Z-axis : 3 digits after the decimal point)

```
G90 G01 ;
  :
N10 X10.000 Z1.000 ;
N20 X10.002 Z1.000 ;
N20 X10.004 Z1.001 ;
  :
```

High precision program command
(Z-axis : 4 digits after the decimal point)

```
G90 G01 ;
  :
N10 X10.000 Z1.0000 ;
N20 X10.002 Z1.0005 ;
N30 X10.004 Z1.0010 ;
  :
```

Program command path) ● is program command point.



- Valid range of various kind data

The valid data range of the maximum stroke and the federate has been not changed by this function.
It is the same as incremental system.

- Command by macro variable

When the macro variable is used for the command value at the axis address where this function is valid, it is analyzed as the command with a decimal point.
The number of decimals is enhanced by 1 digit.

Example) Input unit is metric, and increment system is B (1μm)
In case #500=12.34543
The program command is interpreted as follows.
X[#500] -> X12.3454

- Coordinate screen, program check screen, FOCAS2 and system variables that obtained coordinate value,

When this function is enabled, it is the value of incremental system as same conventional.

- Function that high precision program command is not applied

Functions	Note
Excluding the axis address command	Ex) I, J, K, Q, P, R of circular interpolation, etc…
Sync coefficient of Electronic gear box 2 pair (G81.5)	Gear ration is same as so far
G10 command (Setting command of various data)	same as so far
Dwell command (G04)	same as so far
Waiting function by specifying start point	The judgment position is coordinates of incremental system.
Auxiliary function output in moving axis (G50.9)	It judge by coordinates of incremental system, and the auxiliary function is output.
Stored Stroke Check 2 (G22 X_ Y_ Z_ I_ J_ K_)	As for Coordinate of plus direction (X,Y,Z) the high precision is applied. As for Coordinate of minus direction (I,J,K) the high precision is not applied. (Data of incremental system)
Second auxiliary functions (B)	same as so far
Canned Grinding Cycle (for Grinding machine) (G71 to G75, G77 to G79)	Address A, B command and Grinding range α is same so far

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
10478								HPC

[Input type] Parameter input
[Data type] Bit path

#0 HPC High precision program command function is :
0: Disabled.
1: Enabled.

1.2.3 Specifying the Rotation Axis

Overview

Bit 0 (ROTx) of parameter No.1006 can be used to set each axis to a linear axis or rotary axis. Bit 1 (ROSx) of parameter No. 1006 can be used to select the rotary axis type, A or B, for each axis. See the explanation of the parameters for details of types A and B.
When the roll-over function is used, the values displayed for absolute coordinates are rounded by the shift amount per rotation, as set in parameter No. 1260. This can prevent coordinates for the rotary axis from overflowing. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when bit 2 (RRLx) of parameter No. 1008 is set to 1. The roll-over function is enabled by setting bit 0 (ROAx) of parameter No. 1008 to 1 (for the rotation axis).
For an absolute command, the coordinates after the tool has moved are values rounded by the angle corresponding to one rotation set in parameter No. 1260.
The tool moves in the direction in which the final coordinates are closest when bit 1 (RABx) of parameter No. 1008 is set to 0. For an incremental command, the tool moves the angle specified in the command.

> **NOTE**
> Rotary axis roll–over function cannot be used together with the indexing function (M series) of the index table. To disable the indexing function (M series), set 1 in bit 0 (ITI) of parameter No. 5501 or 0 in bit 3 (IXC) of parameter No. 8132.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx
#1 ROSx Setting linear or rotary axis.

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No.3624)

ROSx	ROTx	Meaning
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values and relative coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No.1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No.3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No.3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

	#7	#6	#5	#4	#3	#2	#1	#0
1008						RRLx	RABx	ROAx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROAx The rotary axis roll-over is
0: Invalid
1: Valid

> **NOTE**
> ROAx specifies the function only for a rotary axis (for which bit 0 (ROTx) of parameter No.1006, is set to 1)

#1 RABx In the absolute commands, the axis rotates in the direction
0: In which the distance to the target is shorter.
1: Specified by the sign of command value.

> **NOTE**
> RABx is valid only when ROAx is 1.

#2 RRLx Relative coordinates are
0: Not rounded by the amount of the shift per one rotation
1: Rounded by the amount of the shift per one rotation

> **NOTE**
> 1 RRLx is valid only when ROAx is 1.
> 2 Assign the amount of the shift per one rotation in parameter No.1260.

1260	The shift amount per one rotation of a rotary axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] Deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

Set the shift amount per one rotation of a rotary axis.
For the rotary axis used for cylindrical interpolation, set the standard value.

	#7	#6	#5	#4	#3	#2	#1	#0
5501								
								ITI

[Input type] Parameter input
[Data type] Bit path

#0 ITI The index table indexing function (M series) is:
0: Enabled.
1: Disabled.

> **NOTE**
> To enable the index table indexing function, set bit 3 (IXC) of parameter No. 8132 to 1 in addition to this parameter. The index table indexing function is enabled only when both ITI and IXC are enabled.

For the rotary axis used for cylindrical interpolation, set the standard value.

	#7	#6	#5	#4	#3	#2	#1	#0
8132								
					IXC			

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#3 IXC Index table indexing function (M series) is:
0: Not Used.
1: Used.

> **NOTE**
> When enabling the index table indexing function, set bit 0 (ITI) of parameter No. 5501 to 0 in addition to this parameter. The index table indexing function is enabled only when both ITI and IXC are enabled.

Note

> **NOTE**
> Rotary axis roll-over function cannot be used together with the indexing function of (M series) the index table.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Rotary axis roll-over function

1.2.4 Controlled Axes Detach

Overview

These signals release the specified control axes from control by the CNC. When attachments are used (such as a detachable rotary table), controlled axis detach signals or bit 7 (RMVx) of parameter No. 0012 are selected according to whether the attachments are mounted. When multiple rotary tables are used in turn, the tables must use motors of the same model.

> ⚠ **WARNING**
> For a vertical axis, in particular, it is necessary to prepare a sequence that starts operating the mechanical brake before the control axis detach operation.
> When this method is applied to a vertical axis, special care should be taken.

Signal

Controlled axis detach signals DTCH1 to DTCH8<Gn124>

[Classification] Input signal

[Function] These signals detach the control axes from control.
These signals are provided for each control axis; the affixed number of the signal name shows the control axis number.

DTCHx

x : 1 ..... The 1st axis is detached.
2 ..... The 2nd axis is detached.
3 ..... The 3rd axis is detached.
: :

[Operation] When the signals are "1", the control unit operates as follows:

<1> Position control is not executed at all. Servo motor excitation is cut.
<2> Servo alarm on the axis is ignored.
<3> Axis interlock signal is assumed to be zero on the detached axis.
<4> A command for automatic or manual operation for the axis does not cause an alarm, but the operation is restrained because the axis interlock signal is "0". In an automatic operation, the execution may stop and hold at the block. Do not execute any command for automatic or manual operation for the axis.
<5> Position display also displays the position of the detached axis.

Controlled axis detach status signals MDTCH1 to MDTCH8<Fn110>

[Classification] Output signal

[Function] These signals notify the PMC that the corresponding axes have been released from control.
These signals are provided for each control axis; the affixed number of the signal name shows the control axis number.
MDTCHx
x : 1 ..... The 1st axis is detached.
2 ..... The 2nd axis is detached.
3 ..... The 3rd axis is detached.
: :

[Output cond.] These signals are "1" in the following case:
- When the corresponding axes are released from control
These signals are "0" in the following case:
- When the corresponding axes are under control

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn124	DTCH8	DTCH7	DTCH6	DTCH5	DTCH4	DTCH3	DTCH2	DTCH1

Fn110	MDTCH8	MDTCH7	MDTCH6	MDTCH5	MDTCH4	MDTCH3	MDTCH2	MDTCH1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0012	RMVx							

[Input type] Setting input
[Data type] Bit axis

#7 RMVx Releasing the assignment of the control axis for each axis
0: Not released
1: Released
(Equivalent to the control axis detachment signals DTCH1, DTCH2, and so forth <G0124>)

> **NOTE**
> 1. RMVx is valid when bit 7 (RMBx) of parameter No. 1005 is set to 1.
> 2. Switch RMVx while the axis is stopping. When switching while the axis is moving, the control axis is detached after the movement of the axis is completed.

	#7	#6	#5	#4	#3	#2	#1	#0
1005	RMBx	MCCx						

[Input type] Parameter input
[Data type] Bit axis

#6 MCCx If a multi-axis amplifier is used, and another axis of the same amplifier is placed in the control axis detach state, the MCC signal of the servo amplifier is:
0: Turned off.
1: Not turned off.

> **NOTE**
> This parameter can be set for a control axis.

#7 RMBx The control axis detachment signal for each axis and the setting input parameter (bit 7 (RMV) of parameter No. 0012) are:

0: Invalid
1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
1818			APDx					

[Input type] Parameter input
[Data type] Bit axis

#5 APDx In the axis with absolute position detector (absolute pulse coder), when the axis is released from state of control axis detach:

0: The parameter APZ (bit 4 of parameter No. 1815) is changed to 0 automatically and the alarm DS0300 is generated.
1: The parameter APZ (bit 4 of parameter No. 1815) remains at 1. After the release of control axis detach, the machine and workpiece coordinate systems are automatically set by using an absolute position detector (absolute pulse coder).

> ⚠ **WARNING**
> 1 It is necessary to detach and attach control axis on the same axis (and the same detector). Do not exchange the axis, etc.
> 2 Do not move an axis of rotary axis (A type) (machine coordinate values are rounded in 0 to 360°.) by more than 180°during control axis detach.
> 3 In case of using the control axis detach function during automatic operation, change controlled axis detach signal DTCH1 to DTCH8 while FIN wait state by using an M code without buffering (parameter No.3411 to No.3432).
> 4 If magnetic pole detection is used, magnetic pole detection is needed when the control axis is reconnected after control axis detach.

Note

> **NOTE**
> 1 Controlled axis detach signals DTCH1 <G124.0>, DTCH2 <G124.1>, DTCH3 <G124.2>, 0 can be changed from “1” to “0” or from “0” to “1” when the power is first turned on or when no movement is being executed along the corresponding axis. If these signals are changed from “0” to “1” when the tool is moving along the corresponding axis, the axis is released from control upon completion of the movement.
> 2 For these signals to be enabled, bit 7 (RMBx) of parameter No. 1005 must be set, indicating the axes are detachable. The signal is invalid for axis without setting.
> 3 Setting bit 7 (RMVx) of parameter No. 0012 from the MDI unit detaches the axes in the same way as these signals.

NOTE

4 Those axes that are released from control lose their reference positions. Reference position return must, therefore, be performed for the axes prior to executing move commands for the axes. Specifying a move command before reference position return has been performed causes alarm PS0224 "ZERO RETURN NOT FINISHED" to be output.
If an axis for which an absolute position detector is used (bit 5 (APCx) of parameter No. 1815 is set to 1) is released from control, the correspondence between the machine position and reference position is lost. Consequently, bit 4 (APZx) of parameter No. 1815 indicating that the correspondence is established is set to 0, resulting in alarm DS0300 "APC ALARM: NEED REF RETURN". After an axis is released from control, perform reference position return to bring the machine position into correspondence with the reference position. But these operations can be omitted by setting bit 5 (APDx) of parameter No.1818 is set to 1.

5 When the control axis detach function is used, however, servo software cannot identify the type of detector in a case where the power to the CNC is turned on with the detector detached from the controlled axis. And an alarm such as a communication alarm may be issued. Such an alarm can be avoided by setting bit 3 (COMSRC) of parameter No. 2017 to 1 "the detector on the semi-closed side is αi/βi pulse coder at all times". Refer to "FANUC AC SERVO MOTOR αi/βi series PARAMETER MANUAL (B-65270EN)" for details.

1.2.5 Outputting the Movement State of an Axis

Overview

The movement state of each axis can be output to the PMC.

Signal

Axis moving signals MV1 to MV8<Fn102>

[Classification] Output signal

[Function] These signals indicate that a control axis is moving.
The signals are provided for each control axis, and the number in the signal name corresponds to the control axis number.

MVx
x : 1 ..... The 1st axis is moving.
2 ..... The 2nd axis is moving.
3 ..... The 3rd axis is moving.
: :
: :

[Output cond.] The signals turn to "1" in the following cases:
- The corresponding axis has started moving.
- In manual handle feed mode, the handle feed axis of the corresponding axis has been elected.

The signals turn to "0" in the following case:
- The corresponding axis has stopped moving and enters the in-position status.

Axis moving direction signals MVD1 to MVD8<Fn106>

[Classification] Output signal

[Function] These signals indicate the movement direction of control axis.
They are provided for each control axis, and the number in the signal name corresponds to the control axis number.

MVDx

x : 1 ..... The moving direction of the 1st axis is minus.
2 ..... The moving direction of the 2nd axis is minus.
3 ..... The moving direction of the 3rd axis is minus.
: :
: :

[Output cond.] "1" indicates the corresponding axes are moving in the minus direction, and "0" indicates they are moving in the plus direction.

> ⚠ **CAUTION**
> These signals maintain their condition during a stop. This signal keeps "1" even if it stops after moving in the minus direction.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn102	MV8	MV7	MV6	MV5	MV4	MV3	MV2	MV1
Fn106	MVD8	MVD7	MVD6	MVD5	MVD4	MVD3	MVD2	MVD1

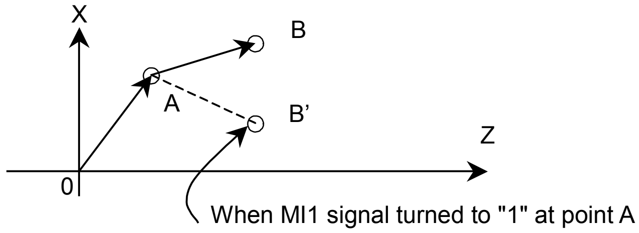
Caution

> ⚠ **CAUTION**
> Axis moving signals and axis moving direction signals are output in both automatic and manual operations.

1.2.6 Mirror Image

Overview

Mirror image can be applied to each axis, either by signals or by parameters (setting input is acceptable). All movement directions are reversed during automatic operation along axes to which a mirror image is applied. Moreover, the absolute coordinate is updated to the same direction as a programmed path, and the machine coordinate and the relative coordinate are updated to the opposite direction from a programmed path.



Mirror image (Example for lathe system)

Fig. 1.2.6 (a)

Mirror image check signals MMI1 to MMI8 <Fn108> indicate whether mirror image is applied to each axis. System variable #3007 contains the same information.
Wrong operation prevention function displays the axis status to the left of the axis name in the display of the machine coordinates, absolute coordinates, relative coordinates, and remaining travel amounts.
Please refer to OPERATOR'S MANUAL (B-64694EN) for details.

Explanation

- Shifting workpiece coordinate system

When operation using mirror image is executed after the workpiece coordinate system is specified, absolute coordinates are updated in the opposite direction with machine coordinates, the workpiece coordinate system is shifted from the machine coordinate system.
This shifted amount can be cancelled by the following operation.

- Manual reference position return
- Workpiece coordinate system preset (G92.1, G50.3)
- Each axis workpiece coordinate system preset signals WPRST1-WPRST8<Gn358>

- Programmable mirror image (G50.1, G51.1)

When programmable mirror image (machining center system) and ordinary mirror image are specified at the same time, programmable mirror image is applied first.

- Retrace

Mirror image is enabled during reverse execution and forward reexecution of the retrace function. Therefore, it is necessary during the reverse execution and the forward reexecution to make mirror image in the same state as the forward execution.

Limitation

- Manual operation and automatic reference position return

Even if mirror image is applied, the following directions are not reversed.

- Direction of manual operation
- Direction of movement, from the intermediate position to the reference position during automatic reference position return (G28)

In these cases, the machine coordinate and the relative coordinate are updated to the same direction as a movement, and the absolute coordinate is updated to the opposite direction from a movement.

M

- Boring cycle (G76 and G87)

Even if mirror image is applied, shift direction for boring cycles (G76 and G87) is not reversed.

- High precision oscillation function

Never attempt to apply mirror image about the oscillation axis.

- Manual linear/circular interpolation

Manual operation of Manual linear/circular interpolation, never use mirror image.

- Program restart, Quick program restart

The tool cannot be returned to a correct position when mirror image is used. However, P type return is possible for a block that switched between ON and OFF most recently or a subsequent block. In this case, mirror image signal status present when the program was interrupted must be maintained.

- Manual intervention and return

When performing manual intervention and return, never use mirror image.

- Tool retract and recover

When retracting the tool manually in the tool withdrawal mode, do not use mirror image.

M

- Single direction positioning

Mirror image is not applied in parameter No.5440 set direction. Even in the mirror image mode, the direction of single direction positioning remains unchanged.

If positioning of linear interpolation type is used, and the state of mirror image when a single direction positioning block is looked ahead differs from the state of mirror image when the execution of the block is started, an alarm is issued. When switching mirror image in the middle of a program, disable looking ahead by specifying a non-buffering M code. Then, switch mirror image when there is no look-ahead block.

- Arbitrary speed threading

Thread cannot be re-machined with mirror image applied.

- General purpose retract

The retract direction is the movement direction of the machine regardless of whether mirror image is valid or not. (A mirror image is not applied to the updating of absolute coordinates.)

- AI contour control II

In the speed control with the cutting feed, the travel direction on the Z-axis is determined with the appropriate NC command. Therefore, if mirror image is applied on the Z-axis, the direction on the Z-axis cannot be determined. When using the speed control with the cutting load, do not use mirror image.

Signal

Mirror image signals MI1 to MI8<Gn106>

[Classification] Input signal

[Function] Select whether to apply a mirror image to each axis.

[Operation] Apply mirror image to those axes for which the signals are "1".

These signals are provided for the controlled axes on a one-to-one basis. A number appended to a signal name represents the controlled axis number.

MIx

x : 1 ..... Applies mirror image to the 1st axis.
2 ..... Applies mirror image to the 2nd axis.
3 ..... Applies mirror image to the 3rd axis.
: :
: :

The mirror image signal can be turned to "1" in the following cases:

(1) During offset cancel;
(2) When the CNC is in the automatic operation stop state and not in the feed hold state.

Mirror image check signals MMI1 to MMI8<Fn108>

[Classification] Output signal

[Function] These signals indicate the mirror image condition of each axis.

The mirror image can be set with the CNC setting data in addition to the input signal from the machine side (PMC).A mirror image is applied by the logical sum of both, and the state is notified to the PMC.

These signals are provided for every control axis; the numeral in the signal name indicates the relevant control axis number.

MMIx

x : 1 ..... Mirror image is applied to the 1st axis
2 ..... Mirror image is applied to the 2nd axis
3 ..... Mirror image is applied to the 3rd axis
: :
: :

[Output cond.] These signals turn to "1" when:

- Mirror image signal MIx of the corresponding axis is 1; or
- Mirror image of the corresponding axis is turned on by setting data.

These signals turn to "0" when:

- Mirror image signal (MIx) of the corresponding axis is 0 and the setting of the mirror image is turned off.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn106	MI8	MI7	MI6	MI5	MI4	MI3	MI2	MI1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn108	MMI8	MMI7	MMI6	MMI5	MMI4	MMI3	MMI2	MMI1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0012								MIRx

[Input type] Setting input
[Data type] Bit axis

#0 MIRx Mirror image for each axis
0: Mirror image is off. (Normal)
1: Mirror image is on. (Mirror)

	#7	#6	#5	#4	#3	#2	#1	#0
3129						MRE		

[Input type] Parameter input
[Data type] Bit path

#2 MRE When mirror image is used, relative coordinates are:
0: Updated with respect to the machine coordinates.
1: Updated with respect to the absolute coordinates.

Set this parameter to 1 when handling relative coordinates in the same way as for the lathe system of the FS16*i*/18*i*/21*i*.

Caution

⚠ **CAUTION**

1 No programmable mirror image (machining center system) affects mirror image check signals MMI1 to MMI8 <Fn108>.
2 Even when the mirror image is applied, commands which do not actuate mirror image (such as automatic reference position return and manual operation) do not affect mirror image check signals MMI1 to MMI8 <Fn108>.

Reference item

Manual name	Item name
OPERATOR'S MANUAL(B-64694EN)	Mirror image
	Operator error prevent functions

1.2.7 Follow-up

Overview

If the machine moves in the state in which position control on controlled axes is disabled (during servo-off, emergency stop, or servo alarm), feedback pulses are accumulated in the error counter. The

CNC reflects the machine movement corresponding to the error count in the current position managed by the CNC. This operation is referred to as follow-up. When follow-up is performed, the current position managed by the CNC does not shift from the actual machine position.
You can select whether to perform follow-up for axes when the servo is turned off.
Follow-up is always performed during emergency stop or a servo alarm.

Explanation

- When follow-up is not performed for the axes for which the servo is turned off

When a follow-up signal *FLWU <Gn007.5> is “1” or a bit 0 (FUPx) of parameter No.1819 is 1, follow-up is not performed. The error is added to the error counter as a servo error.
In this case, the machine moves to compensate for the error when the servo off signal SVF1 to SVF8 <Gn126> changes to “0”.
In general, follow-up is not used if the machine is mechanically clamped when position control is disabled for the controlled axes.

- When follow-up is performed for the axes for which the servo is turned off

When a bit 0 (FUPx) of parameter No.1819 is 0 and a follow-up signal *FLWU is “0”, the follow-up function is performed.
That is, it is considered that there is a command corresponding to the movement of the machine, and update the present position of the CNC so that the error counter to zero. In this case, even if the servo-off signal is restored to "0", the position of the machine remains dislocated. However, since the present position of the CNC is moving corresponding to the actual position of the machine, so, if it executes the program with cycle start at the state, it moves to the correct position with the first absolute command. In general, follow-up is used with a mechanical handle (a method of moving a motor by a mechanical handle mechanism and moving a machine).

⚠ CAUTION

1 If, during automatic operation, a servo off signal is issued with the setting that causes follow-up to be performed (a bit 0 (FUPx) of parameter No.1819 = 0 and a follow-up signal *FLWU<Gn007.5> = “0”), even if the machine is moved with external force or other means, the travel distance will not immediately reflected in the program coordinate system. Until it is reflected, the machine path will be shifted by the amount of movement due to the external force as in the Fig. 1.2.7 (a) .

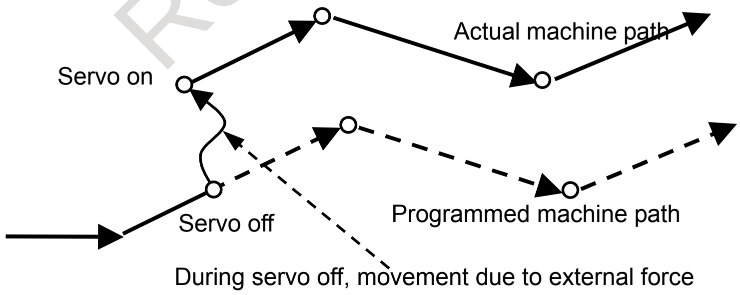


Fig. 1.2.7 (a)

The following method is available to reflect the amount of movement during servo off in the program coordinate system. If not wishing to shift the path, be sure to follow the procedure below to adjust the coordinates and execute an absolute command.

- Exit from automatic operation with a reset, single block stop, or feed hold, and then make a restart.

> ⚠ **CAUTION**
> 2 If a servo off signal is issued with the setting that does not cause follow-up to be performed (a bit 0 (FUPx) of parameter No.1819 = 1 or a follow-up signal *FLWU<Gn007.5> = "1"), even if the machine is moved with external force or other means, the machine will be retracted by the travel distance in the servo on state and, therefore, the path will never shift in subsequent automatic operation. The amount of movement due to the external force in the servo off state is regarded as a servo positional deviation and is stored inside the CNC. Thus, when a servo on signal is issued, axis moving occurs to cancel this servo positional deviation. The machine moves at a speed in accordance with the servo loop gain and if the amount is large, this may give the machine a shock.

Signal

Follow-up signal *FLWU<Gn007.5>

[Classification] Input signal
[Function] Select whether to perform follow-up when the servo is turned off for those axes for which bit 0 (FUPx) of parameter No.1819 is 0.
[Operation] 0: Performs follow-up.
1: Does not perform follow-up.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn007			*FLWU					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1819								FUPx

[Input type] Parameter input
[Data type] Bit axis

#0 FUPx To perform follow-up when the servo is off is set for each axis.
0: The follow-up signal, *FLWU, determines whether follow-up is performed or not.
When *FLWU is "0", follow-up is performed.
When *FLWU is "1", follow-up is not performed.
1: Follow-up is not performed.

> **NOTE**
> When using the index table indexing function, set FUPx to 1 for a control axis subject to index table indexing.

Caution

> ⚠ **CAUTION**
> Following operation is disabled for the axis in follow-up.
> - Manual handle interruption

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (this manual)	Servo off/mechanical handle feed

1.2.8 Servo off/Mechanical Handle Feed

Overview

Place the controlled axes in the servo off state, stop the current to the servo motor, which disables position control. However, the position detection feature functions continuously, so the current position is not lost.

These signals are used to prevent the servo motors from overloading when the tools on the axes are mechanically clamped under certain machining conditions on the machine, or to move the machine by driving the motors by mechanical handles.

Please refer to “Follow-up” for details of follow-up in the servo off state.

Signal

Servo off signals SVF1 to SVF8<Gn126>

[Classification] Input signal

[Function] Select whether to place each axis in the servo off state.
These signals are provided for the controlled axes on a single axis basis. A number appended to a signal represents a controlled axis number.

SVFx

x : 1 ..... Servo off for the first axis
2 ..... Servo off for the second axis
3 ..... Servo off for the third axis
: :

[Operation] These signals put the axes for which the signals are “1” in the servo off state (the current to the servo motor is stopped). This disables position control. However, the position detection feature continues to function, so the current position is not lost.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn126	SVF8	SVF7	SVF6	SVF5	SVF4	SVF3	SVF2	SVF1

Caution

⚠ **CAUTION**

1 In general, interlock is applied to an axis while the servo off signal for that axis is "1".

2 The mechanical clamp is done by using the auxiliary function. Set the timing for the auxiliary function, mechanical clamp and servo off signals as shown in the Fig. 1.2.8 (a). The clamp command auxiliary function should be executed only after the distribution end signal DEN <Fn001.3> turned to "1".

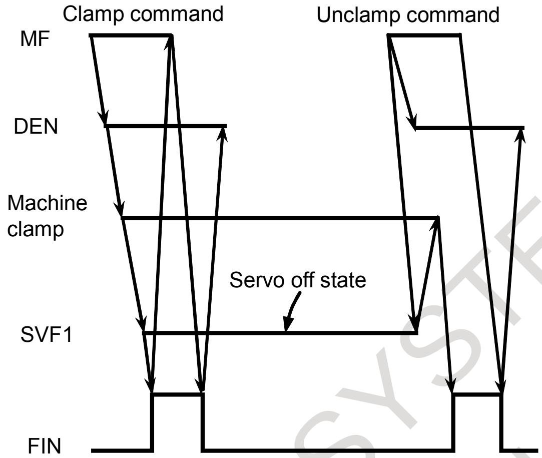


Fig. 1.2.8 (a)

3 If, during automatic operation, a servo off signal is issued with the setting that causes follow-up to be performed (a bit 0 (FUPx) of parameter No.1819 = 0 and a follow-up signal *FLWU<Gn007.5> = "0"), even if the machine is moved with external force or other means, the travel distance will not immediately reflected in the program coordinate system. Until it is reflected, the machine path will be shifted by the amount of movement due to the external force as in the Fig. 1.2.8 (b).

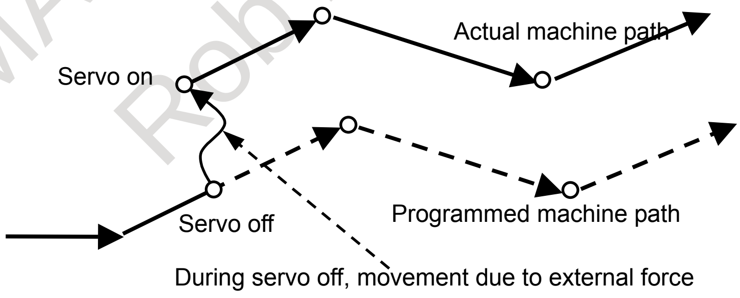


Fig. 1.2.8 (b)

The following method is available to reflect the amount of movement during servo off in the program coordinate system. If not wishing to shift the path, be sure to follow the procedure below to adjust the coordinates and execute an absolute command.

- Exit from automatic operation with a reset, single block stop, or feed hold, and then make a restart.

> ⚠ **CAUTION**
> 4 If a servo off signal is issued with the setting that does not cause follow-up to be performed (a bit 0 (FUPx) of parameter No.1819 = 1 or a follow-up signal *FLWU<Gn007.5> = "1"), even if the machine is moved with external force or other means, the machine will be retracted by the travel distance in the servo on state and, therefore, the path will never shift in subsequent automatic operation. The amount of movement due to the external force in the servo off state is regarded as a servo positional deviation and is stored inside the CNC. Thus, when a servo on signal is issued, axis moving occurs to cancel this servo positional deviation. The machine moves at a speed in accordance with the servo loop gain and if the amount is large, this may give the machine a shock.
> 5 If the positioning deviation exceeds the value set in the parameter No.1830 in servo-off state, alarm SV0410, "EXCESS ERROR (STOP)" is occurred. If the parameter No.1830 is set to 0, the positioning deviation is not checked in servo-off state.

Parameter

1830	Axis-by-axis positional deviation limit at servo-off time

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

This parameter is used to set a positional deviation limit at servo-off time, on an axis-by-axis basis.
If the value specified with this parameter is exceeded at servo-off time, a servo alarm is issued to cause an immediate stop (same as an emergency stop). Usually, set the same value as a positional deviation limit at stop time.

> **NOTE**
> If this parameter is set to 0, the positioning deviation is not checked in servo-off state.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (this manual)	Follow-up

1.2.9 Position Switch

Overview

Position switch signals can be output to the PMC while the machine coordinates along a controlled axes are within a specified ranges.
Using parameters, specify arbitrary controlled axes and machine coordinate operating ranges for which position switch signals PSW01 to PSW16 <Fn070,Fn071> are output.
Up to 10 position switch signals can be output.
Up to 16 points can be expanded by setting bit 1 (EPW) of parameter No. 6901 to 1.

> ⚠ **CAUTION**
> The position switch function is enabled after reference position return is completed.

Signal

Position switch signals PSW01 to PSW16<Fn070, Fn071>

[Classification] Output signal

[Function] Indicates that the machine coordinates along the controlled axes specified by parameters Nos. 6910 to 6925 are within the ranges specified by parameters Nos. 6930 to 6945 and 6950 to 6965.
The position switch signal corresponding to the n-th position switch function is PSWn.
(n : 1 to 16)

[Output cond.] These signals are “1” in the following case:

- When the machine coordinates along the controlled axes are within the specified ranges.

These signals are “0” in the following case:

- When the machine coordinates along the controlled axes are not within the specified ranges.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn070	PSW08	PSW07	PSW06	PSW05	PSW04	PSW03	PSW02	PSW01
Fn071	PSW16	PSW15	PSW14	PSW13	PSW12	PSW11	PSW10	PSW09

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6901						PSA	EPW	

[Input type] Parameter input
[Data type] Bit path

#1 EPW The number of position switches is:
0: Not extended.
1: Extended.

#2 PSA In determination of a position switch function operation range, a servo delay amount (positional deviation) and a delay amount in acceleration/deceleration control are:
0: Not considered.
1: Considered.

6910	Controlled axis for which the 1-st position switch function is performed (PSWA01)
to	to
6925	Controlled axis for which the 16-th position switch function is performed (PSWA16)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the controlled axis number (intra-path relative axis number) corresponding to one of the first to sixteenth position switch functions. When the machine coordinate of the corresponding axis is within a parameter-set range, the corresponding position switch signal is output to the PMC.

> **NOTE**
> The setting of 0 means that the position switch function of the number is not used.

6930	**Maximum value of the operating range of the 1-st position switch (PSW101)**
to	**to**
6945	**Maximum value of the operating range of the 16-th position switch (PSW116)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the maximum value of the operating range of the first to sixteenth position switches.

> **NOTE**
> 1 For a diameter-specified axis, use diameter values to specify the parameters used to set the maximum and minimum values of an operating range.
> 2 The position switch function is enabled upon completion of reference position return.
> 3 Please do not specify the range that steps over 360° for the rotation axis.

6950	**Minimum value of the operating range of the 1-st position switch (PSW201)**
to	**to**
6965	**Minimum value of the operating range of the 16-th position switch (PSW216)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the minimum value of the operating range of the first to sixteenth position switches.

> **NOTE**
> 1 For a diameter-specified axis, use diameter values to specify the parameters used to set the maximum and minimum values of an operating range.
> 2 The position switch function is enabled upon completion of reference position return.
> 3 Please do not specify the range that steps over 360° for the rotation axis.

1.2.10 High-Speed Position Switch

Overview

The high-speed position switch function monitors the current position at shorter intervals than the normal position switch function to output a high-speed precise position switch signal.

In the same way as for the normal position switch function, using parameters, specify arbitrary controlled axes and machine coordinate operating ranges for which position switch signals are output.
Up to six high-speed position signals can be output. Up to 16 points can be expanded by setting bit 7 (HPE) of parameter No. 8500 to 1.

> ⚠ **CAUTION**
> The high-speed position switch function is enabled after reference position return is completed.

Explanation

- Output addresses of high-speed position switch signals

High-speed position switch signals are output to the PMC Y signal addresses set using parameter No. 8565. If a nonexistent address is set for the parameter, the high-speed position switch function is disabled. If you do not want to use the PMC Y signal addresses, you can set bit 0 (HPF) of parameter No. 8501 to 1 to use high-speed position switch signals as normal output signals (using F signal addresses).

> ⚠ **WARNING**
> Never use the Y signal used on the PMC ladder. In case of using it with mistakes, the machine may perform unexpected operation.

Signal

High-speed position switch signals HPS01 to HPS16<Yxxx,Yxxx+1><Fn293,Fn294>

[Classification] Output signal

[Function] Indicates that the machine coordinates along the controlled axes specified by parameters Nos. 8570 to 8579 and 12201 to 12206 are within the ranges specified by parameters Nos. 8580 to 8589, 12221 to 12226, 8590 to 8599, and 12241 to 12246.
The position switch signal corresponding to the n-th position switch function is HPSn.
(n : 1 to 16)

[Output cond.] These signals are "1" in the following case:
- When the machine coordinate value along the controlled axis is within a specified range.

These signals are "0" in the following case:
- When the machine coordinate value the along the controlled axis is not within a specified range.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Yxxx	HPS08	HPS07	HPS06	HPS05	HPS04	HPS03	HPS02	HPS01
Yxxx+1	HPS16	HPS15	HPS14	HPS13	HPS12	HPS11	HPS10	HPS09

xxx indicates the address set using parameter No. 8565.
When bit 0 (HPF) of parameter No. 8501 is set to 1, the signal addresses are F293 and F294. (Y signal addresses are not used.)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8500	HPE							

[Input type] Parameter input
[Data type] Bit path

#7 HPE The maximum number of high-speed position switches is:
0: 6.

1: 16.

	#7	#6	#5	#4	#3	#2	#1	#0
8501							HPS	HPF

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 HPF The output signal of a high-speed position switch is output to:
0: Address Y.
1: Address F.

#1 HPS The current position used with the high-speed position switch:
0: Considers a servo error.
1: Does not consider a servo error.

	#7	#6	#5	#4	#3	#2	#1	#0
8504	E08	E07	E06	E05	E04	E03	E02	E01
8505	E16	E15	E14	E13	E12	E11	E10	E09

[Input type] Parameter input
[Data type] Bit path

E01 to E16 These parameters specify whether to enable or disable each corresponding high-speed position switch.
The Table 1.2.10 (a) shows the correspondence between the bits and switches.
The settings of each bit have the following meaning:
0: The switch corresponding to the bit is enabled.
1: The switch corresponding to the bit is disabled (always outputs 0).

Table 1.2.10 (a)

Parameter	Switch
E01	1st high-speed position switch
E02	2nd high-speed position switch
E03	3rd high-speed position switch
:	:
E16	16th high-speed position switch

8565	Output address of the high-speed position switch signal

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 126

This parameter sets a Y signal address to which the high-speed position switch signal is output. The Y signal addresses consisting of the value set in this parameter and the set value plus 1 are used.
If a nonexistent address is set, the high-speed position switch function is disabled. When bit 0 (HPF) of parameter No. 8501 is set to 1, however, this parameter has no effect.

Table 1.2.10 (b) Relationship between the high-speed position switches and the addresses to be output

	Signal address to be output	Controlled axes number	Maximum operation range	Minimum operation range
1st to 8th	“Value set in the parameter No.8565”	8570 to 8577	8580 to 8587	8590 to 8597
9th to 16th	“Value set in the parameter No.8565” + 1	8578 to 8579, 12201 to 12206	8588 to 8589, 12221 to 12226	8598 to 8599, 12241 to 12246

> ⚠ **WARNING**
> 1 Be sure not to use any Y signal already used in the PMC ladder with this function. If used, the machine may behave in an unexpected manner.
> 2 If you want to use high-speed position switches for multiple paths, use a different Y signal output address for each path.

> ⚠ **CAUTION**
> 1 Specifying a nonexistent signal address causes the high-speed position switch function to be disabled.
> 2 Y signal address Y127 cannot be specified for this function.
> 3 Address output signals (Y1001 and above) on the M-NET board cannot be specified for this function.
> 4 When "0" is set to the parameter, it is output to Y0 and Y1. Please set the available address when the high-speed position switch is used.

8570	**Controlled axis for which the first high-speed position switch function is performed**
to	**to**
8579	**Controlled axis for which the tenth high-speed position switch function is performed**

12201	**Controlled axis for which the eleventh high-speed position switch function is performed**
to	**to**
12206	**Controlled axis for which the sixteenth high-speed position switch function is performed**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes

Each of these parameters sets a controlled axis number for which each of the first to sixteenth high-speed position switch functions is performed.
Set 0 for the number corresponding to a high-speed position switch which is not to be used.

NOTE
Parameters Nos. 8576 to 8579 and 12201 to 12206 are valid only when bit 7 (HPE) of parameter No. 8500 is 1.

8580	Maximum value of the operation range of the first high-speed position switch
to	to
8589	Maximum value of the operation range of the tenth high-speed position switch

12221	Maximum value of the operation range of the eleventh high-speed position switch
to	to
12226	Maximum value of the operation range of the sixteenth high-speed position switch

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Each of these parameters sets the maximum value of the operation range of each of the first to sixteenth high-speed position switches. If such a setting that maximum value < minimum value is made, no operation range exists, so that the high-speed position switch does not operate.

NOTE
Parameters Nos. 8586 to 8589 and 12221 to 12226 are valid only when bit 7 (HPE) of parameter No. 8500 is 1.

8590	Minimum value of the operation range of the first high-speed position switch
to	to
8599	Minimum value of the operation range of the tenth high-speed position switch

12241	Minimum value of the operation range of the eleventh high-speed position switch
to	to
12246	Minimum value of the operation range of the sixteenth high-speed position switch

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Each of these parameters sets the minimum value of the operation range of each of the first to sixteenth high-speed position switches. If such a setting that maximum value < minimum value is made, no operation range exists, so that the high-speed position switch does not operate.

NOTE
Parameters Nos. 8596 to 8599 and 12241 to 12246 are valid only when bit 7 (HPE) of parameter No. 8500 is 1.

1.2.11 Axis Total Travel Distance Display

Overview

Total travel distance of axis is calculated, the distance of each axis is displayed on the diagnosis screen. If target distance is set, signal indicates that the travel distance exceeds the target.

Explanation

- Display of total travel distance

Total travel distance of axis is displayed on diagnosis data Nos. 365 and 366. When bit 3 (ETD) of parameter No. 8906 is 1, this function is valid.
Diagnosis data No. 365 becomes 0 when total travel distance clear signal TDCx<Gn726> is set to "1" from "0".
Diagnosis data No. 366 becomes 0 when it exceeds the upper limit.
When bit 3 (ETD) of parameter No. 8906 is 0, this function is executed as follows:

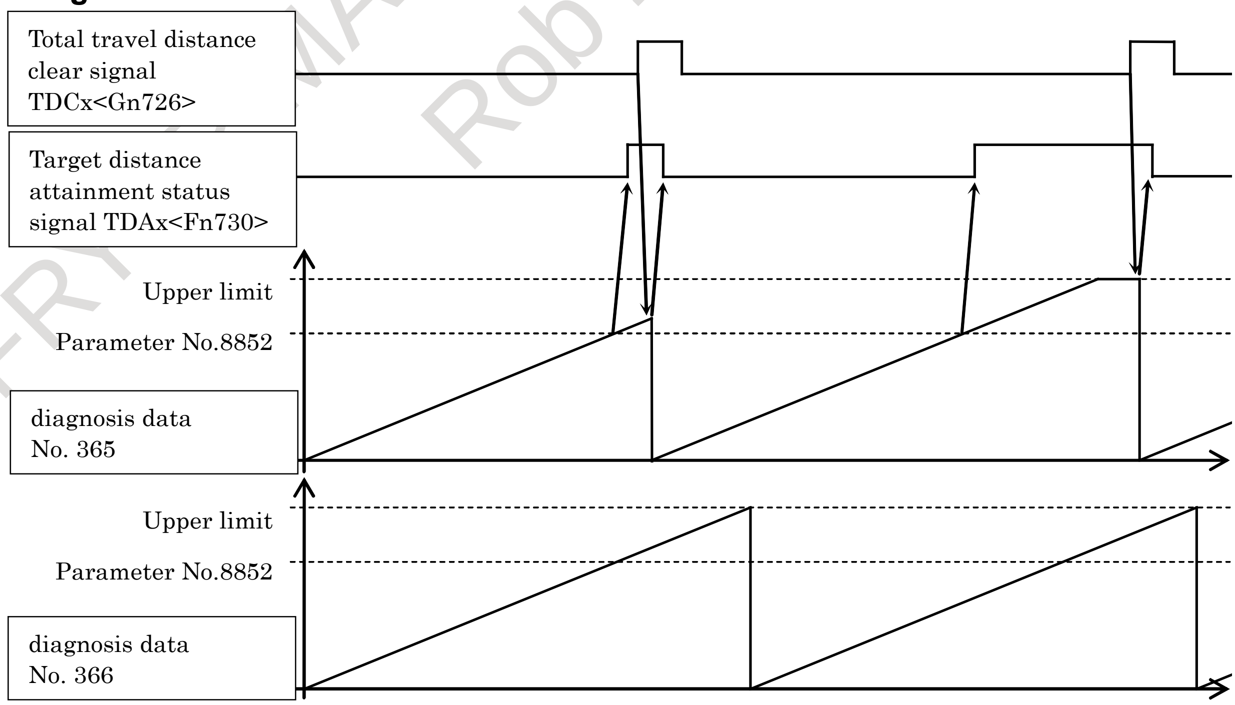
- Total travel distance is not calculated. However, the value calculated in the past is displayed.
- Diagnosis data No. 365 is not cleared by total travel distance clear signal TDCx<Gn726>.

	Total travel distance clear signal TDCx<Gn726> is set to "1" from "0"	Travel distance exceeds the upper limit.
Diagnosis data No. 365	Becomes 0	Keeps the upper limit
Diagnosis data No. 366	Doesn't become 0	Becomes 0

- Indication of target distance attainment

When total travel distance (diagnosis data No. 365) exceeds the target distance set in parameter No. 8852, target distance attainment status signal TDAx<Fn730> becomes "1". Even if travel distance exceeds target, the accumulation of diagnostic data(No.365) continues.
When bit 3 (ETD) of parameter No. 8906 is 1, this function is valid. If bit 3 (ETD) of parameter No. 8906 is 0 or the value set in parameter No.8852 is outside the data range, target distance attainment status signal TDAx<Fn730> always becomes 0.

Timing chart



NOTE
1 Total travel distance is not calculated at the following command:
- Feedrate control of PMC axis control of when bit 2 (VCP) of parameter No. 8007 is 1 and bit 4 (EVP) of parameter No. 8005 is 0.
- Torque control of PMC axis control of when follow-up is not performed (bit 4 (TQF) of parameter No. 1803 is 0).

2 Total travel distance of axis to which servo is invalid (bit 4 (KSVx) of parameter No. 11802) is not calculated.
3 In the axis during servo-off with follow-up (bit 0 (FUPx) of parameter No.1819 is 0 and follow-up signal *FLWU<Gn007.5> is “0”), the travel distance by the external force is added to total travel distance.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8906					ETD			

[Input type] Parameter input
[Data type] Bit

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#3 ETD Axis total travel distance display is:
0: Invalid.
1: Valid.

8852	Target distance of each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] m, 100 inch, 360 deg
[Valid data range] 1 to 999999999

When diagnosis data No. 365 exceeds the value set in this parameter, target distance attainment status signal TDAx<Fn730> become "1".

NOTE
1 In case of rotation axis, unit of data is 360 degree. In case of linear axis, unit of data is m or 100 inch according to the least command increment.
2 If the value set in this parameter is outside the data range or bit 3 (ETD) of parameter No. 8906 is 0, target distance attainment status signal TDAx<Fn730> always becomes 0.
3 This parameter is specified as radius specification. Diameter/radius specification does not affect this parameter.

Signal

Total travel distance clear signal TDC1 to TDC8<Gn726>

[Classification] Input signal

[Function] Total travel distance (diagnosis data No. 365) of the corresponding axis becomes 0.

[Operation] When these signals are set to "1" from "0", total travel distance (diagnosis data No. 365) of the corresponding axis becomes 0.

> **NOTE**
> 1 When bit 3 (ETD) of parameter No. 8906 is 1, these signals are valid. If bit 3 (ETD) of parameter No. 8906 is 0, diagnosis data No. 365 doesn't become 0.
> 2 Set these signals to "0" after confirming either of the following conditions:
> - Target distance attainment status signal TDAx<Fn730> becomes “0” from “1”
> - Diagnosis data No. 365 becomes 0
> - 128 msec or more have passed since these signals became “1”

Target distance attainment status signal TDA1 to TDA8<Fn730>

[Classification] Output signal

[Function] These signals indicate that the travel distance exceeds the target.

[Output cond.] These signals become “1” when:

- Total travel distance (diagnosis data No. 365) exceeds the target distance set in parameter No. 8852.

These signals become “0” when:

- Total travel distance (diagnosis data No. 365) doesn’t exceed the target distance set in parameter No. 8852.

> **NOTE**
> When bit 3 (ETD) of parameter No. 8906 is 1, these signals are valid. If bit 3 (ETD) of parameter No. 8906 is 0 or the value set in parameter No.8852 is outside the data range, these signals always become 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn726	TDC8	TDC7	TDC6	TDC5	TDC4	TDC3	TDC2	TDC1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn730	TDA8	TDA7	TDA6	TDA5	TDA4	TDA3	TDA2	TDA1

Diagnosis data

365	Total travel distance (clearable)
366	Total travel distance

[Data type] 2-word axis
[Unit of data] m, 100 inch, 360 deg
[Valid data range] 0 to 42949672 (When unit of data is m)
0 to 16909320 (When unit of data is 100 inch)
0 to 4294967295 (When unit of data is 360 deg)

Total travel distance of axis is displayed.
When total travel distance clear signal TDCx<Gn726> is set to "1" from "0", diagnosis data No. 365 becomes 0. Diagnosis data No. 366 becomes 0 when it exceeds the upper limit.

	Total travel distance clear signal TDCx<Gn726> is set to "1" from "0"	Travel distance exceeds the upper limit.
Diagnosis data No. 365	Becomes 0	Keeps the upper limit
Diagnosis data No. 366	Doesn't become 0	Becomes 0

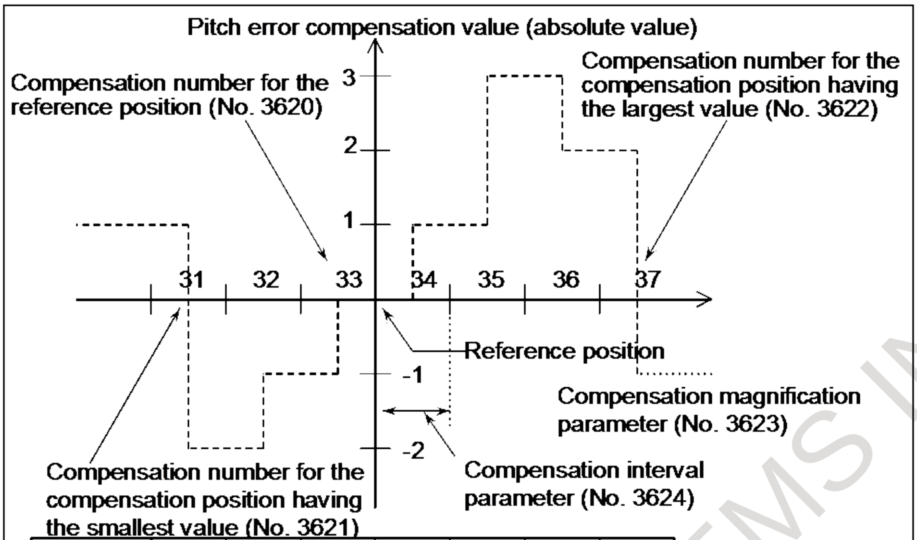
NOTE
1 In case of rotation axis, unit of data is 360 degree. In case of linear axis, unit of data is m or 100 inch according to the least command increment.
2 If bit 3 (ETD) of parameter No. 8906 is 1, total travel distance is calculated.
3 If bit 3 (ETD) of parameter No. 8906 is 0, total travel distance is not calculated. However, the value calculated in the past is displayed.
4 These diagnosis data are displayed as radius specification. Diameter/radius specification does not affect these diagnosis data.

1.3 ERROR COMPENSATION

1.3.1 Stored Pitch Error Compensation

Overview

If pitch error compensation data is specified, pitch errors of each axis can be compensated in detection units per axis.
Pitch error compensation data is set for each compensation position at the intervals specified for each axis. The origin of compensation is the reference position to which the tool is returned.
Pitch error compensation data can be set with external devices such as the Handy File (see OPERATOR'S MANUAL). Compensation data can also be set directly with the MDI unit.
The following parameters must be set for pitch error compensation. Set the pitch error compensation value for each pitch error compensation position number set by these parameters.
In the following example of Fig. 1.3.1 (a), 33 is set for the pitch error compensation number at the reference position.



Compensation position number	31	32	33	34	35	36	37
Compensation value to be set	-3	+1	+1	+1	+2	-1	-3

Fig. 1.3.1 (a)

- Pitch error compensation position at the reference position (for each axis): Parameter No.3620
- Pitch error compensation position having the smallest value (for each axis): Parameter No.3621
- Pitch error compensation position having the largest value (for each axis): Parameter No.3622
- Pitch error compensation magnification (for each axis): Parameter No.3623
- Interval of the pitch error compensation positions (for each axis): Parameter No.3624

Explanation

- Specifying the compensation position

To assign the compensation positions for each axis, specify the positive direction or the negative direction relative to the compensation position No. of the reference position. If the machine stroke exceeds the specified range on either the positive direction or the negative direction, the pitch error compensation does not apply beyond the range.

- Compensation position number

1536 compensation positions from No. 0 to 1535 are available on the pitch error setting screen. Assign arbitrary positions for each axis using parameters.
The number of the compensation position at the reference position (parameter No.3620), number of the compensation position having the smallest value (parameter No.3621), and number of the compensation position having the largest value (parameter No.3622) must be set for each axis.
The name of each axis is displayed before the smallest compensation position number on the pitch error setting screen.

- Interval of compensation positions

The pitch error compensation positions are equally spaced to parameter No. 3624. Set the space between two adjacent positions for each axis.
The minimum interval between pitch error compensation positions is limited and obtained from the following equation:
Minimum interval of pitch error compensation positions = maximum feedrate (rapid traverse rate)/7500
Unit :

Minimum interval of pitch error compensation positions: mm, inches, deg.
Maximum feed rate: mm/min, inch/min, deg/min

[Example]

When the maximum rapid traverse rate is 15000 mm/min, the minimum interval between pitch error compensation positions is 2 mm.

Example

- For linear axis

- Machine stroke: -400 mm to +800 mm
- Interval between the pitch error compensation positions: 50 mm
- No. of the compensation position of the reference position: 40

If the above is specified, the No. of the farthest compensation position in the negative direction is as follows:

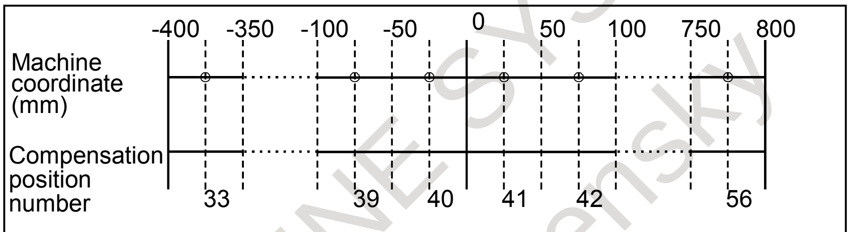
No. of the compensation position of the reference position - (Machine stroke on the negative side/Interval between the compensation positions) + 1= 40 - 400/50 + 1= 33

No. of the farthest compensation position in the positive direction is as follows:

No. of the compensation position of the reference position + (Machine stroke on the positive side/Interval between the compensation positions)= 40 + 800/50= 56

A fraction of the division of machine stroke on the negative/positive side by interval between the compensation positions is raised up to an integer.

The correspondence between the machine coordinate and the compensation position No. is as Fig. 1.3.1(b):



Compensation values are output at the positions indicated by ○.

Fig. 1.3.1 (b)

Therefore, set the parameters as Table 1.3.1 (a):

Table 1.3.1 (a)

Parameter	Setting value
3620 : Compensation number for the reference position	40
3621 : Smallest compensation position number	33
3622 : Largest compensation position number	56
3623 : Compensation magnification	1
3624 : Interval between pitch error compensation positions	50.0

The compensation amount is output at the compensation position No. corresponding to each section between the coordinates.

The Fig. 1.3.1 (c) is an example of the compensation amounts.

Number	33	34	35	36	37	38	39	40	41	42	43	44	45	46	47	48	49	...	56
Compensation value	-2	-1	-1	+2	0	+1	0	+1	+2	+1	0	-1	-1	-2	0	+1	+2	...	1

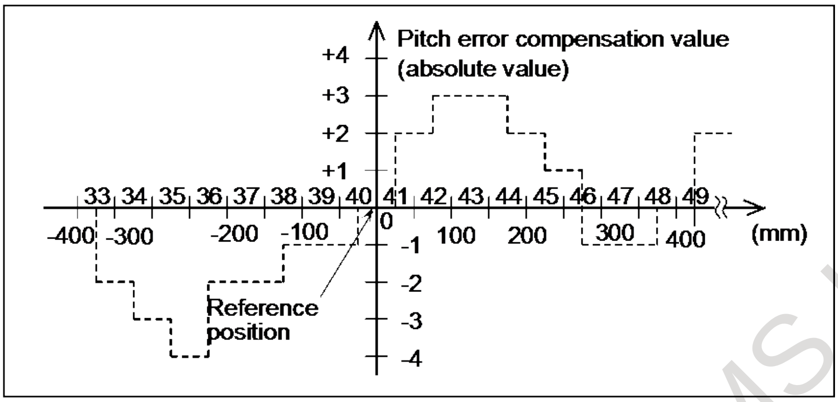


Fig. 1.3.1 (c)

- For rotary axis

- Amount of movement per rotation: 360°
- Interval between pitch error compensation positions: 45°
- No. of the compensation position of the reference position: 60

In the above case, the number of the most distance compensation position on the - side is equal to the number of the compensation position of the reference position + 1 = 60 + 1 = 61 for a rotary axis.
The No. of the farthest compensation position in the positive direction is as follows:
No. of the compensation position of the reference position + (Move amount per rotation/Interval between the compensation positions)= 60 + 360/45= 68
The correspondence between the machine coordinate and the compensation position No. is as Fig. 1.3.1 (d):

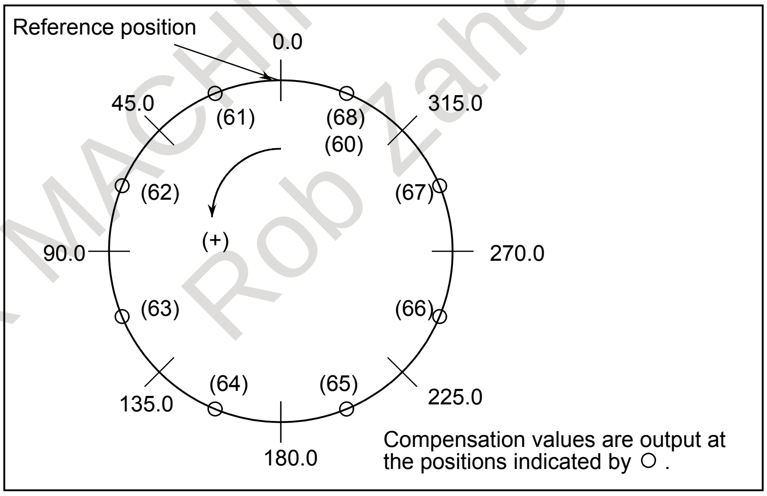


Fig. 1.3.1 (d)

Therefore, set the parameters as Table 1.3.1 (b):

Table 1.3.1 (b)

Parameter	Setting value
3620 : Compensation number for the reference position	60
3621 : Smallest compensation position number	61
3622 : Largest compensation position number	68
3623 : Compensation magnification	1
3624 : Interval between pitch error compensation positions	45.0
3625 : Movement value per rotation	360.0

If the sum of the compensation values for positions 61 to 68 is not 0, pitch error compensation values are accumulated for each rotation, causing positional deviation.
The same value must be set for compensation positions 60 and 68.

The Fig. 1.3.1 (e) is an example of compensation amounts.

Number	60	61	62	63	64	65	66	67	68
Compensation value	+1	-2	+1	+3	-1	-1	-3	+2	+1

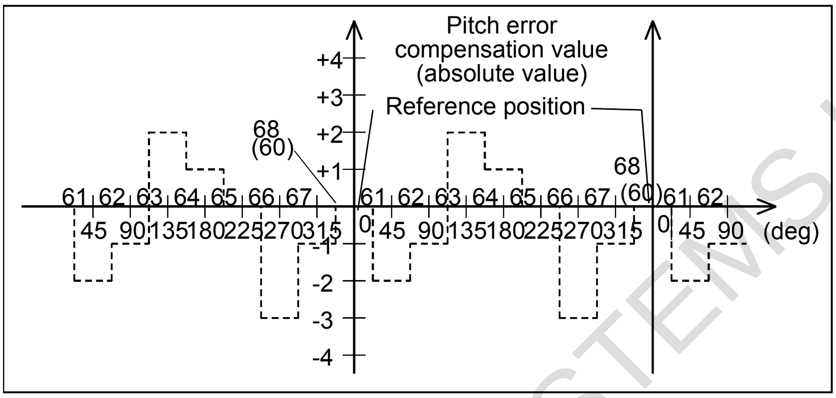


Fig. 1.3.1 (e)

Procedure for displaying and setting the pitch error compensation data

Procedure

1 Set the following parameters:

- Number of the pitch error compensation point at the reference position (for each axis) : Parameter No. 3620
- Number of the pitch error compensation point having the smallest value (for each axis) : Parameter No. 3621
- Number of the pitch error compensation point having the largest value (for each axis) : Parameter No. 3622
- Pitch error compensation magnification (for each axis) : Parameter No. 3623
- Interval of the pitch error compensation points (for each axis) : Parameter No. 3624
- Travel distance per revolution of pitch error compensation of the rotary axis type (for each axis) : Parameter No. 3625

When using bi-directional pitch error compensation (setting bit 0 (BDPx) of parameter No. 3605 to 1), specify the following parameters in addition to the pitch error compensation parameter.

- Number of the pitch error compensation point at the negative end (for travel in the positive direction, for each axis) : Parameter No. 3621
- Number of the pitch error compensation point at the positive end (for travel in the positive direction, for each axis) : Parameter No. 3622
- Number of the pitch error compensation point at the negative end (for travel in the negative direction, for each axis) : Parameter No. 3626
- Pitch error compensation in the reference position when moving to the reference position from opposite to the reference position return direction (for each axis) : Parameter No. 3627

2 Press function key SYSTEM.

3 When the display unit is 10.4-inch, press the continuous menu key ▷, then press chapter selection soft key [PITCH ERROR]. The following screen is displayed:

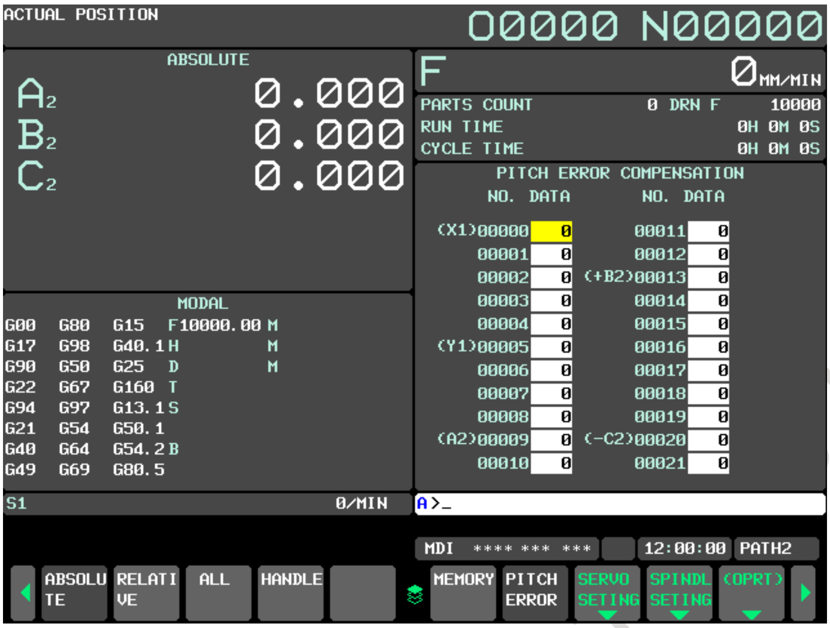


Fig. 1.3.1 (f) PITCH ERROR COMPENSATION screen (10.4-inch display unit)

3 When the display unit is 15/19-inch, press vertical soft key [PITCH ERROR]. The following screen is displayed:

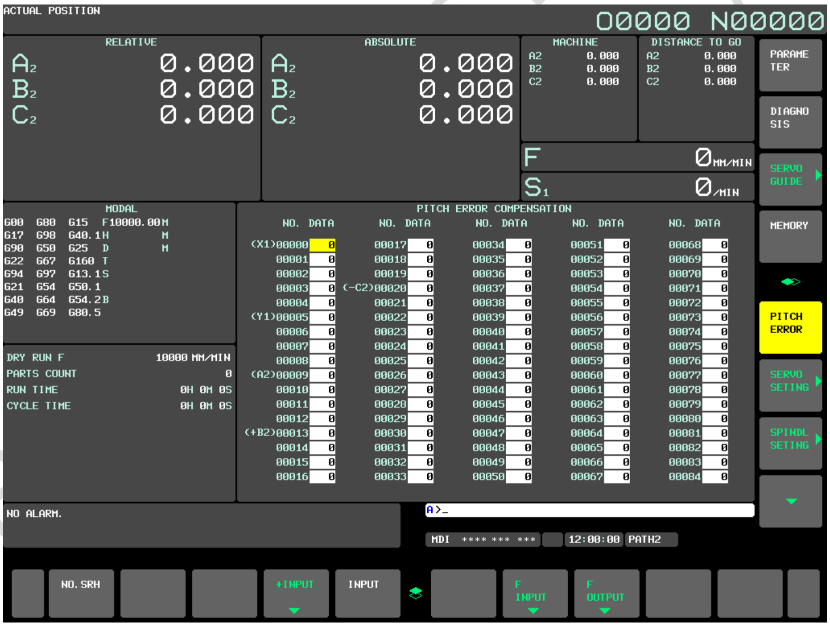
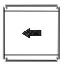
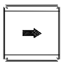


Fig. 1.3.1 (g) PITCH ERROR COMPENSATION screen (15-inch display unit)

4 Move the cursor to the compensation point number to be set in either of the following ways:
- Enter the compensation point number and press the soft key [NO.SRH].
- Move the cursor to the compensation point number using the page keys,  and , and cursor keys, , , , and .

5 Enter a value with numeric keys and press soft key [INPUT].

If bit 5 (PAD) of parameter No. 11350 is 1, an axis name is displayed next to the compensation point number set in parameter No. 3621 for determining the most negative pitch error compensation point number. Also, if the bi-directional pitch error compensation function is enabled, "+ axis name" is displayed next to the compensation point number set in parameter No. 3621 for setting the compensation point during movement in the positive direction, and "- axis name" is displayed next to the compensation point number set in parameter No. 3626 for setting the compensation point during movement in the negative direction.

NOTE
1 If the setting of the pitch error compensation parameter is not correct, the axis name of that axis is not displayed.
2 For a rotation axis, an axis name is displayed next to the pitch error compensation point number of the reference position set in parameter No. 3620.

Parameter

3620	Number of the pitch error compensation position for the reference position for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1535

Set the number of the pitch error compensation position for the reference position for each axis.

3621	Number of the pitch error compensation position at extremely negative position for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1535

Set the number of the pitch error compensation position at the extremely negative position for each axis.

3622	Number of the pitch error compensation position at extremely positive position for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1535

Set the number of the pitch error compensation position at the extremely positive position for each axis.
This value must be larger than set value of parameter No.3620.

3623	Magnification for pitch error compensation for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 100

Set the magnification for pitch error compensation for each axis.
If the magnification is set to 1, the same unit as the detection unit is used for the compensation data.
If 0 is set, compensation is not performed.

3624	Interval between pitch error compensation positions for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] See the description below.

The pitch error compensation positions are arranged with equal spacing. The space between two adjacent positions is set for each axis. The minimum interval between pitch error compensation positions is limited and obtained from the following equation:
Minimum interval between pitch error compensation positions = maximum feedrate/7500
Unit : mm, inch, deg or mm/min, inch/min, deg/min

[Example] When the maximum feedrate is 15000 mm/min, the minimum interval between pitch error compensation positions is 2 mm.

3625	Travel distance per revolution in pitch error compensation of rotary axis type

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] See the description below.

If the pitch error compensation of rotary axis type is performed (bit 1 (ROSx) of parameter No. 1006 is set to 0 and bit 0 (ROTx) of parameter No. 1006 is set to 1), set the travel distance per revolution. The travel distance per revolution does not have to be 360 degrees, and a cycle of pitch error compensation of rotary axis type can be set.
However, the travel distance per revolution, compensation interval, and number of compensation points must satisfy the following condition:

(Travel distance per revolution)
= (Compensation interval) × (Number of compensation points)

The compensation at each compensation point must be set so that the total compensation per revolution equals 0.

> **NOTE**
> If 0 is set, the travel distance per revolution becomes 360 degrees.

Warning

> ⚠ **WARNING**
> - **Compensation value range**
>   Compensation values can be set within the range from -127 × compensation magnification (detection unit) to +127 × compensation magnification (detection unit). The compensation magnification can be set for each axis within the range from 0 to 100 in parameter 3623.
> - **Pitch error compensation of the rotary axis**
>   For the rotating axis, the interval between the pitch error compensation positions shall be set to one per integer of the amount of movement (normally 360°) per rotation. The sum of all pitch error compensation amounts per rotation must be made to 0. Also, set the same compensation value to a position and the same position with one rotation.
> - **Conditions where pitch error compensation is not performed**
>   Note that the pitch error is not compensated in the following cases:
>   - When the machine is not returned to the reference position after turning on the power. This excludes the case where an absolute position detector is employed.
>   - If the interval between the pitch error compensation positions is 0.
>   - If the compensation position Nos. on the positive or negative direction do not fall within the range of 0 to 1535.
>   - For linear axis, if the compensation position Nos. do not conform to the following relationship:
>   Negative side ≤ Reference position < Positive side

Note

> **NOTE**
> For multipath control, axes that have the same axis name but that have different paths must use different compensation position Nos.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Inputting Pitch Error Compensation Data
	Outputting Pitch Error Compensation Data

1.3.2 Backlash Compensation

Overview

- Backlash compensation

Function for compensating for lost motion on the machine. Set a compensation value in parameter No. 1851, in detection units from 0 to ±9999 pulses for each axis.

- Backlash compensation for each rapid traverse and cutting feed

When the bit 4 (RBK) of parameter No.1800 is set to 1, more precise machining can be performed by changing the backlash compensating value depending on the feedrate, the rapid traverse or the cutting feed.

Let the measured backlash at cutting feed be A and the measured backlash at rapid traverse be B. The backlash compensating value is shown Table 1.3.2 (a) depending on the change of feedrate (cutting feed or rapid traverse) and the change of the direction of movement.

Table 1.3.2 (a)

Change of feedrate / Change of direction of movement	Cutting feed to cutting feed	Rapid traverse to rapid traverse	Rapid traverse to cutting feed	Cutting feed to rapid traverse
Same direction	0	0	±α	±(-α)
Opposite direction	±A	±B	±(B+α)	±(B+α)

- $\alpha = (A-B) / 2$ (However, if α is not an integer, it is $\alpha = (A-(B-1)) / 2$.)
- The positive or negative direction for compensating values is the direction of movement.

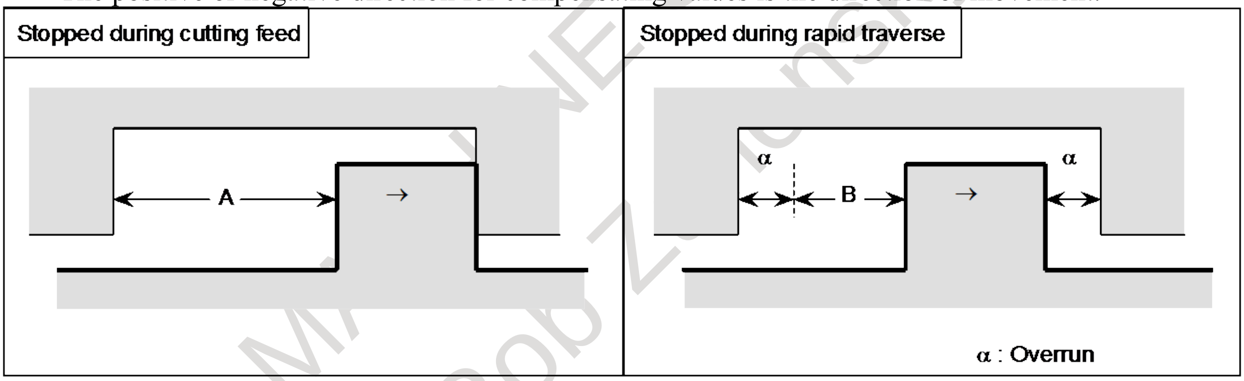


Fig. 1.3.2 (a)

- Assign the measured backlash at cutting feed (A) in parameter No. 1851 and that at rapid traverse (B) in parameter No. 1852.

- Tolerance for reverse pulse of the axis moving direction

The output of the backlash compensation pulse is controlled, from reversing of the movement direction of the axis to multiplying the movement value (unit of detection) that set by the parameter (No.1853).

As a result, the output of the backlash compensation pulse to the movement minutely repeated can be controlled.

And, the tolerance parameter (No.1853) for reverse pulse of the axis moving direction is also effective in the following functions.

- Backlash compensation for each rapid traverse and cutting feed
  (Backlash of the same direction is excluded)
- Smooth backlash compensation

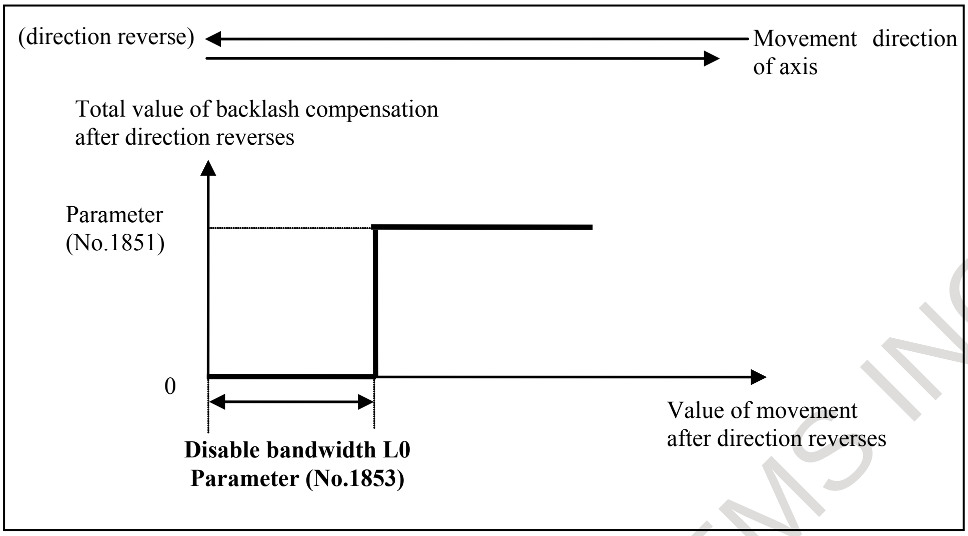


Fig. 1.3.2 (b) Tolerance for reverse pulse of the axis moving direction

> **NOTE**
> When an excessive value is input to the parameter (No.1853) of this function, the backlash compensation for each rapid traverse and cutting feed output of the same direction is done.
> When the backlash must be invalided, please adjust compensation value parameter (No.1848,1851,1852) to 0.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1800				RBK				

[Input type] Parameter input
[Data type] Bit path

#4 RBK Backlash compensation applied separately for rapid traverse and cutting feed
0: Not performed
1: Performed

	#7	#6	#5	#4	#3	#2	#1	#0
1802				BKL15x				

[Input type] Parameter input
[Data type] Bit axis

#4 BKL15x When the direction of a movement is determined in backlash compensation:
0: The compensation amount is not considered.
1: The compensation amount (pitch error, straightness, external machine coordinate system shift, etc.) is considered.

1851	Backlash compensating value for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -9999 to 9999
Set the backlash compensating value for each axis.

When the machine moves in a direction opposite to the reference position return direction after the power is turned on, the first backlash compensation is performed.

1852	Backlash compensating value used for rapid traverse for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -9999 to 9999

Set the backlash compensating value used in rapid traverse for each axis. (This parameter is valid when bit 4 (RBK) of parameter No. 1800 is set to 1.) More precise machining can be performed by changing the backlash compensating value depending on the feedrate, the cutting feed or the rapid traverse positioning. Let the measured backlash at cutting feed be A and the measured backlash at rapid traverse be B. The backlash compensating value is shown Table 1.3.2 (b) depending on the change of feedrate (cutting feed or rapid traverse) and the change of the direction of movement.

Table 1.3.2 (b)

Change of feedrate / Change of direction of movement	Cutting feed to cutting feed	Rapid traverse to rapid traverse	Rapid traverse to cutting feed	Cutting feed to rapid traverse
Same direction	0	0	$\pm\alpha$	$\pm(-\alpha)$
Opposite direction	$\pm$A	$\pm$B	$\pm(B+\alpha)$	$\pm(B+\alpha)$

NOTE
1 α=(A-B)/2 (However, if α is not an integer, it is α= (A-(B-1)) / 2.)
2 The positive or negative direction for compensating values is the direction of movement.

1853	Tolerance for reverse pulse of axis moving direction of backlash compensation

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

The tolerance value for reverse pulse of the axis moving direction of the backlash compensation is set.
After the value set by the parameter at time when the direction of the movement changed is moved, the backlash compensation pulse is output.
When this parameter set to 0, the function of invalid bandwidth is disabled, and the backlash compensation is done when the direction of the movement is changed.

Caution

⚠ **CAUTION**
The backlash compensation for each rapid traverse and cutting feed is not performed until the first reference position return is completed after the power is turned on. Under this state, the normal backlash compensation is performed according to the value specified in parameter No. 1851 irrespective of a rapid traverse or a cutting feed.

Note

NOTE

1. When backlash compensation is applied separately for cutting feed and rapid traverse, jog feed is regarded as cutting feed.
2. If parameter No.1851 is changed when the backlash compensation or the backlash compensation for each rapid traverse and cutting feed is performed; the new compensation value becomes valid when the machine moves in a direction opposite to the reference return direction.
3. If parameter No.1852 is changed when the backlash compensation for each rapid traverse and cutting feed is performed; the new compensation value becomes valid when the machine moves at rapid traverse regardless of direction.

1.3.3 Smooth Backlash

Explanation

With normal backlash compensation, all backlash compensating value are output at the location where the direction of axis moving reverses. (Fig. 1.3.3 (a))

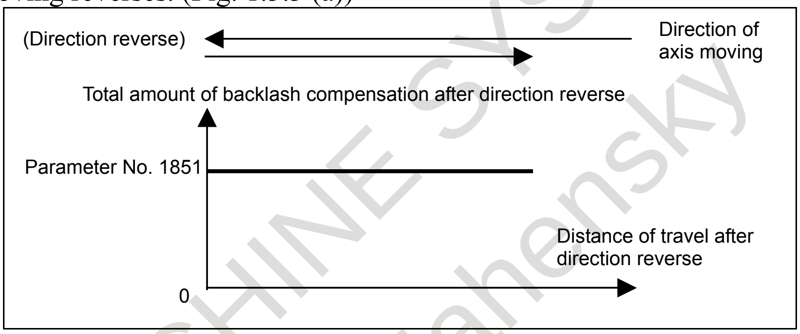


Fig. 1.3.3 (a) Normal backlash compensation

With smooth backlash compensation, backlash compensating value are output in accordance with the distance from the location where the direction of axis moving reverses, so that fine backlash compensation corresponding to the characteristics of the machine is possible. (Fig. 1.3.3 (b))

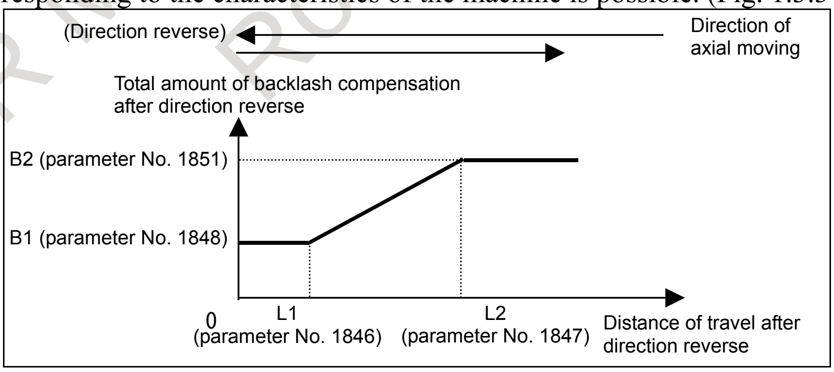


Fig. 1.3.3 (b) Smooth backlash compensation

To enable this function set SBL, bit 2 of parameter No. 1817, to 1.

- First stage backlash compensation output

At the location where the direction of axis moving reverses, the first stage backlash compensation output is performed. Set the first stage backlash compensation B_1, using parameter No.1848.

- Second stage backlash compensation output

At the point the tool moves by the distance L1 from the location where the direction of axis moving reverses, the second stage backlash compensation output is started. And, at the point the tool moves by the distance L2 from the location where the direction of axis moving reverses, the second stage backlash compensation output is terminated. The total amount of backlash at the stage where the second stage backlash compensation output is terminated, or B2, is the same as the backlash compensation set using parameter No.1851. Set the distances L1 and L2, using parameters Nos. 1846 and 1847, respectively. The rate of increase of the second stage backlash compensation output is calculated from the following equation.

$$\textit{Rate of increase of second backlash compensation output} = \frac{\textit{Parameter}\,\text{No.}1851 - B_1}{L_2 - L_1} \qquad (1)$$

If backlash compensation for each rapid traverse and cutting feed is enabled (RBK, bit 4 of parameter No. 1800 = 1), the total amount of backlash compensation at the stage where the second stage backlash compensation output is terminated, or B2, is the backlash compensation as determined by parameters Nos. 1852 and 1851, the reversed direction, and the rapid traverse/cutting feed mode. The rate of increase of the second stage backlash compensation output remains the same as that during cutting. (Expression 1)
The following shows an example in which the tool is changed from cutting feed to rapid traverse feed and the direction reverses. (Fig. 1.3.3 (c))

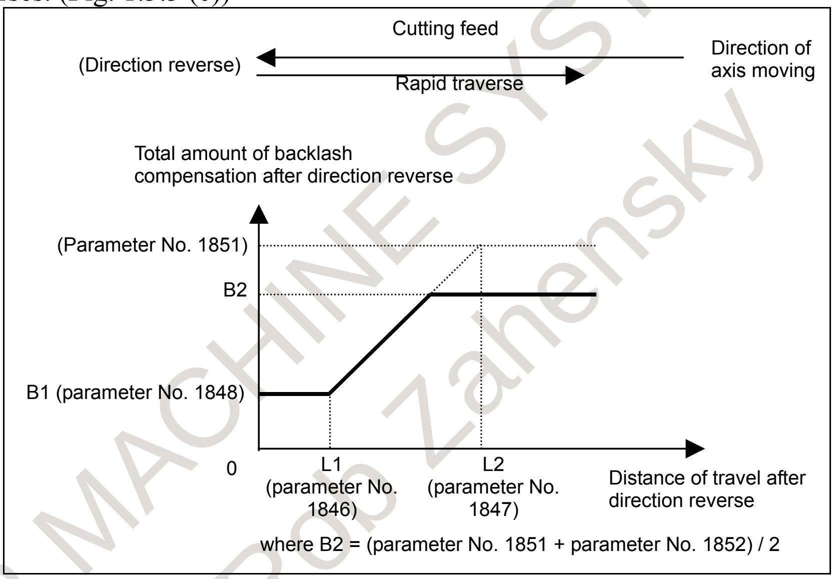


Fig. 1.3.3 (c) In the case of a change from cutting feed to rapid traverse

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1800				RBK				

[Input type] Parameter input
[Data type] Bit path

#4 RBK Backlash compensation applied separately for cutting feed and rapid traverse
0: Not performed
1: Performed

	#7	#6	#5	#4	#3	#2	#1	#0
1817						SBL		

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#2 SBL Smooth backlash compensation is :
0: Disabled.
1: Enabled.

1846	Distance for starting the second stage of smooth backlash compensation

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

For each axis, set the distance from the point where the axis movement direction is reversed to the point where the second stage of smooth backlash compensation is started.

If the following condition is not satisfied, smooth backlash compensation is disabled:
Value of parameter No. 1846 ≥ 0
Value of parameter No. 1846 < value of parameter No. 1847

1847	Distance for ending the second stage of smooth backlash compensation

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

For each axis, set the distance from the point where the axis movement direction is reversed to the point where the second stage of smooth backlash compensation is ended.

If the following condition is not satisfied, smooth backlash compensation is disabled:
Value of parameter No. 1846 ≥ 0
Value of parameter No. 1846 < value of parameter No. 1847

1848	Value of the first stage of smooth backlash compensation

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -9999 to 9999

Set the value of the first stage of smooth backlash compensation for each axis.
If the setting of this parameter is larger than the total backlash compensation value, smooth backlash compensation is not performed.
When a negative value is set for the backlash compensating value for each axis (parameter No. 1851), set a negative value in this parameter. If the sign set in this parameter is different from that set for the backlash compensating value for each axis (parameter No. 1851), compensation is performed, assuming that the value of the first stage of smooth backlash compensation is 0.

1851	Backlash compensating value for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -9999 to 9999

Set the backlash compensating value for each axis.
When the machine moves in a direction opposite to the reference position return direction after the power is turned on, the first backlash compensation is performed.

1852	Backlash compensating value used for rapid traverse for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -9999 to 9999

Set the backlash compensating value used in rapid traverse for each axis. (This parameter is valid when bit 4 (RBK) of parameter No. 1800 is set to 1.) More precise machining can be performed by changing the backlash compensating value depending on the feedrate, the cutting feed or the rapid traverse positioning. Let the measured backlash at cutting feed be A and the measured backlash at rapid traverse be B. The backlash compensating value is shown below depending on the change of feedrate (cutting feed or rapid traverse) and the change of the direction of movement.

Change of feedrate / Change of direction of movement	Cutting feed to cutting feed	Rapid traverse to rapid traverse	Rapid traverse to cutting feed	Cutting feed to rapid traverse
Same direction	0	0	$\pm\alpha$	$\pm(-\alpha)$
Opposite direction	$\pm$A	$\pm$B	$\pm(B+\alpha)$	$\pm(B+\alpha)$

NOTE
1 $\alpha=(A-B)/2$
2 The positive or negative direction for compensating values is the direction of movement.

	#7	#6	#5	#4	#3	#2	#1	#0
11601		SBN						

[Input type] Parameter input
[Data type] Bit

#6 SBN When the dual position feedback or the monitoring semi-full error is used in servo function, the smooth backlash compensation is executed :
0: According to the setting of bit 4 of parameter No.2206 and bit 5 of parameter No.2010.
1: In the semi-closed loop side.

1.3.4 Straightness Compensation

Overview

For a machine tool with a long stroke, deviations in straightness between axes may affect the machining accuracy. For this reason, when an axis moves, other axes are compensated in detection units to improve straightness. This improvement results in better machining accuracy.
When an axis (parameters Nos. 5711 to 5716) moves, the corresponding compensation axis (parameters Nos. 5721 to 5726) is compensated.
That is, the compensation axis is compensated at the pitch error compensation position (See Subsection “Stored Pitch Error Compensation”) of the moving axis.

- Relation between pitch error compensation points and straightness compensation

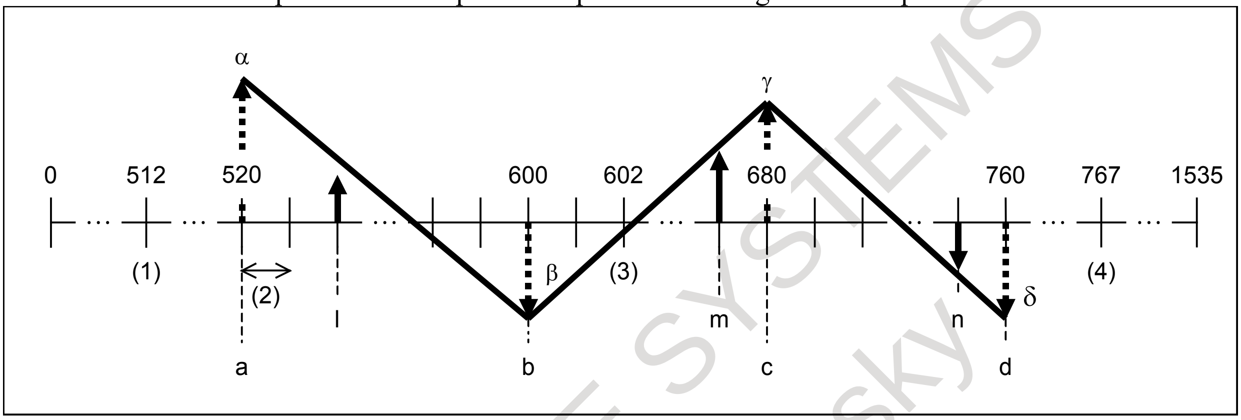


Fig. 1.3.4 (a)

To perform straightness compensation, stored pitch error compensation must be set for the moving axis.
(1) Number of the most distant pitch error compensation point on the - side (parameter No.3621)
(2) Pitch error compensation point interval (parameter No. 3624)
(3) Number of the pitch error compensation point of the reference position (parameter No. 3620)
(4) Number of the most distant pitch error compensation point on the + side (parameter No. 3622)
In Fig. 1.3.4 (a), (1), (3) and (4) are 512, 602 and 767, respectively.
Straightness compensation parameters must be set.
a,b,c,d : Compensation point numbers. (parameters Nos. 5731 to 5754, Nos. 13301 to 13324)
α,β,γ,δ : Compensation amounts at compensation points a, b, c and d (parameters Nos. 5761 to 5784, Nos. 13351 to 13374)
In Fig. 1.3.4 (a), a, b, c and d are 520, 600, 680 and 760, respectively.
Unlike stored pitch error compensation, whose amount is set up for an individual compensation point, an amount of straightness compensation is calculated for individual compensation points by setting up four typical points and compensation amounts for them.
Example:
In above figure, the compensation amount at the individual compensation point is as follows
At compensation point l between compensation point a and point b:

$\alpha + (\beta-\alpha) / (b-a) \times (l-a)$

At compensation point m between compensation point b and point c:

$\beta + (\gamma-\beta) / (c-b) \times (m-b)$

At compensation point n between compensation point c and point d:

$\gamma + (\delta-\gamma) / (d-c) \times (n-c)$

Example

Imagine a table whose Y-axis ball screw is placed on its X-axis ball screw. If the X-axis ball screw is inclined at a certain angle because of, for example, bending, the machining precision related to the Y-axis becomes low because its ball screw is affected by the gradient of the X-axis ball screw. (Left figure shown Fig. 1.3.4 (b))

Specifying the X-axis and Y-axis, respectively, as a moving axis and a compensation axis by means of straightness compensation causes the Y-axis (compensation axis) position to be compensated according to the X-axis (moving axis) position, thus increasing the machining precision. (Right figure shown Fig. 1.3.4 (b))

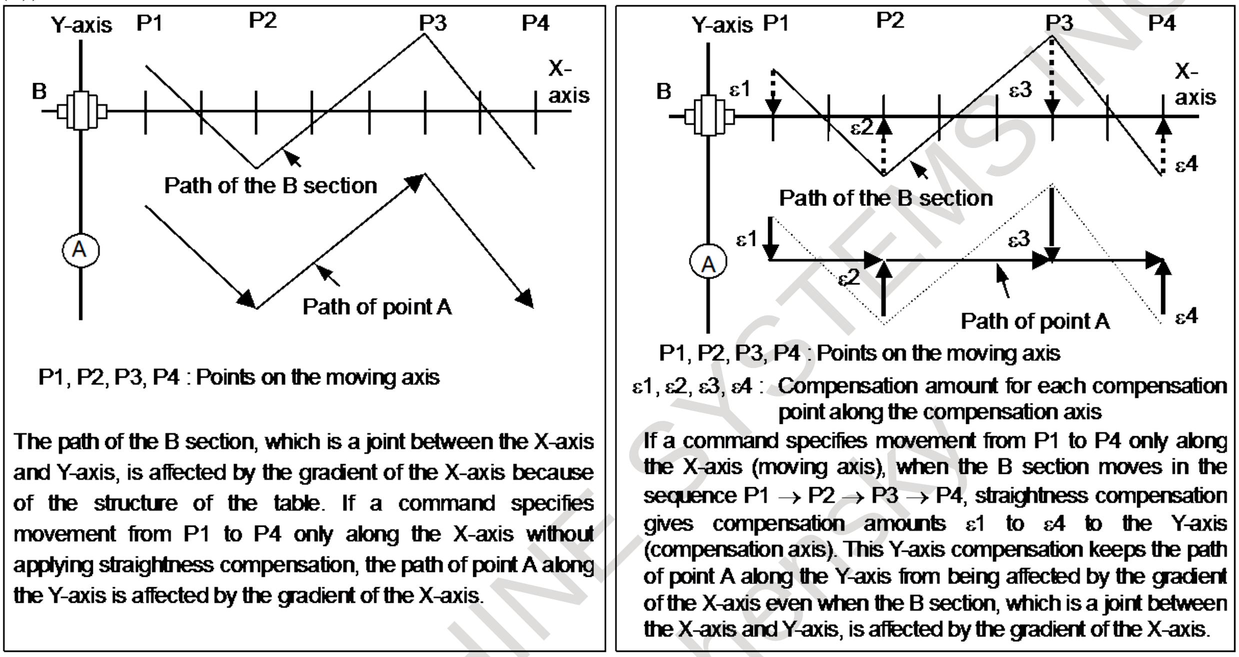


Fig. 1.3.4 (b)

- Function without turning off the power changing parameter of straightness compensation value

The compensation value of straightness compensation on each compensation point (parameter No. 5761 to 5784, No.13351 to 13374) can be changed without turning off the power by G10, pmc-window or MDI input.

The rewritten data is reflected from the following compensation point.

This function is activated when bit 0 (RSR) of parameter No.5700 is set to 1.

NOTE

1. When all axes have stopped, the parameters of the compensation value can be changed.
2. Please change the data of the interpolated straightness compensation (No.6000 to 9071 of pitch error setting screen) on the reference point (compensation value = 0).
3. When an illegal value is set to the compensation value parameter in each compensation point, the alarm DS5046"ILLEGAL PARAMETER (S-COMP.)" is generated.
   At this time, when the movement axis is moved by the manual operation method etc, the straightness compensation becomes invalid only the axis that sets an illegal value.

(1) Operation when the compensation value parameter is changed
When the compensation value parameter in each compensation point (a to d point) of the straightness compensation is changed, the changed value is reflected from the next pitch error compensation point.

Example)
The compensation point number (parameter No.5731 to 5733) of movement axis 1 is set to "a", "b" and "c". And, the value (parameter No.5761 to 5763) of the compensation in each compensation point is set to "α", "β" and " γ".
When the value "β" of the compensation of movement axis 1 is changed to "β' " and "γ" is changed to "γ' ", the value of the compensation becomes like the solid line of Fig. 1.3.4 (c).

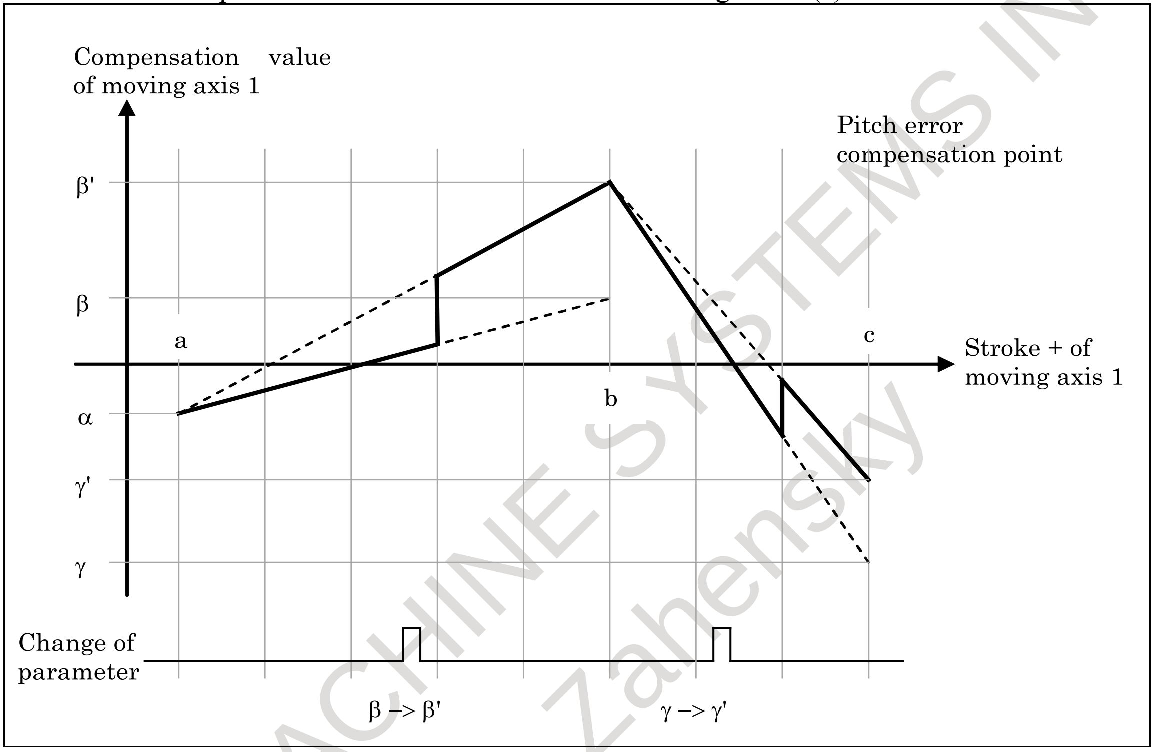


Fig. 1.3.4 (c) Example

(2) The value of compensation in reference point
The compensation value of reference point is outputed based on the setting of the parameter.

Example)
The compensation point number (parameter No.5731 to 5733) of movement axis 1 is set to "a", "b" and "c". And, the value (paremter No.5761 to 5763) of the compensation in each compensation point is set to "α", "β" and " γ".
However, the compensation point number "a" is set to be a compensation point number of the reference point. At this time, it becomes like the solid line of Fig. 1.3.4 (d).

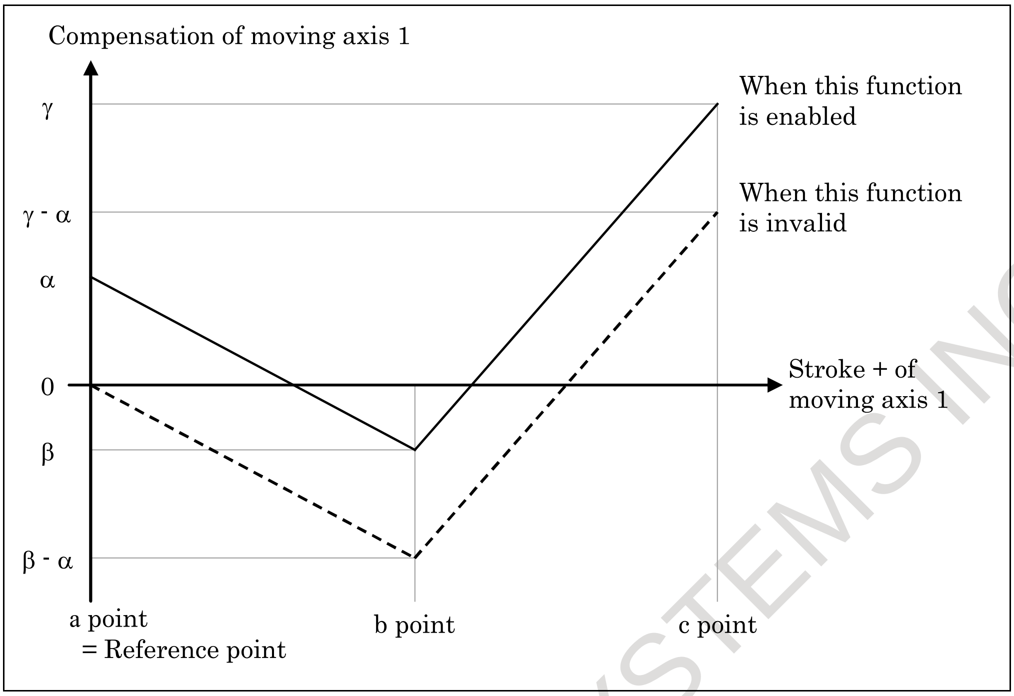


Fig. 1.3.4 (d) Example

> **NOTE**
> When this function is invalid, the value of the compensation becomes 0 in the reference point. (Dotted line of Fig. 1.3.4 (d))

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5700								RSR

[Input type] Parameter input
[Data type] Bit path

#0 RSR When the compensation value data (parameter No.5761 to 5784, No.13351 to 13374) of the straightness compensation is set, the power off/on is :
0: Required.
1: Not required.

> **NOTE**
> 1 When this parameter is set, the power must be turned off before operation is continued.
> 2 When this function is enabled and the compensation value of straightness compensation on each compensation point (parameter No. 5761 to 5784, No.13351 to 13374) is changed, the alarm of turning off the power is not generated.

5711	Straightness compensation : Axis number of moving axis 1

5712	Straightness compensation : Axis number of moving axis 2

5713	Straightness compensation : Axis number of moving axis 3

5714	Straightness compensation : Axis number of moving axis 4

5715	Straightness compensation : Axis number of moving axis 5
5716	Straightness compensation : Axis number of moving axis 6

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes
Set the axis number of a moving axis in straight compensation.
When 0 is set, compensation is not performed.

5721	Straightness compensation : Axis number of compensation axis 1 for moving axis 1
5722	Straightness compensation : Axis number of compensation axis 2 for moving axis 2
5723	Straightness compensation : Axis number of compensation axis 3 for moving axis 3
5724	Straightness compensation : Axis number of compensation axis 4 for moving axis 4
5725	Straightness compensation : Axis number of compensation axis 5 for moving axis 5
5726	Straightness compensation : Axis number of compensation axis 6 for moving axis 6

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

5731	Straightness compensation : Compensation point number a of moving axis 1
to	to
5734	Straightness compensation : Compensation point number d of moving axis 1
5741	Straightness compensation : Compensation point number a of moving axis 2
to	to
5744	Straightness compensation : Compensation point number d of moving axis 2
5751	Straightness compensation : Compensation point number a of moving axis 3
to	to
5754	Straightness compensation : Compensation point number d of moving axis 3
13301	Straightness compensation : Compensation point number a of moving axis 4
to	to
13304	Straightness compensation : Compensation point number d of moving axis 4
13311	Straightness compensation : Compensation point number a of moving axis 5
to	to
13314	Straightness compensation : Compensation point number d of moving axis 5
13321	Straightness compensation : Compensation point number a of moving axis 6
to	to
13324	Straightness compensation : Compensation point number d of moving axis 6

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 1535
These parameters set compensation point numbers in stored pitch error compensation. Set four compensation points for each moving axis.

5761	Compensation corresponding compensation point number a of moving axis 1
to	to
5764	Compensation corresponding compensation point number d of moving axis 1
5771	Compensation corresponding compensation point number a of moving axis 2
to	to
5774	Compensation corresponding compensation point number d of moving axis 2
5781	Compensation corresponding compensation point number a of moving axis 3
to	to
5784	Compensation corresponding compensation point number d of moving axis 3
13351	Compensation corresponding compensation point number a of moving axis 4
to	to
13354	Compensation corresponding compensation point number d of moving axis 4
13361	Compensation corresponding compensation point number a of moving axis 5
to	to
13364	Compensation corresponding compensation point number d of moving axis 5
13371	Compensation corresponding compensation point number a of moving axis 6
to	to
13374	Compensation corresponding compensation point number d of moving axis 6

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Unit of data] Detection unit
[Valid data range] -32767 to 32767
Each of these parameters sets a compensation value for each moving axis compensation point.

Alarm and message

Number	Message	Description
PW1103	ILLEGAL PARAMETER (S-COMP.128)	The parameter for setting 128 straightness compensation points or the parameter compensation data is incorrect,
PW5046	ILLEGAL PARAMETER (S-COMP.)	The parameter for setting straightness compensation is incorrect.
DS5046	ILLEGAL PARAMETER (S-COMP.)	The parameter for setting straightness compensation is incorrect.

Note

NOTE

1. To use this function, the stored pitch error compensation is required.
2. The straightness compensation function can be used after a moving axis and its compensation axis have returned to the reference position.
3. After setting parameters for straightness compensation, be sure to turn off the NC power. (When bit 0 (RSR) of parameter No.5700 is set to 1, the part of parameter is excluded.)
4. Set parameters for straightness compensation according to the following conditions:
   - The difference of compensation at a compensation point must be within the range -127 to 127. The difference of compensation at a compensation point between compensation point a and point b is (β-α) / (b-a).
   - Compensation positions must be set so that "a≤b≤c≤d" is satisfied.
   - Compensation positions must exist between the compensation position with the largest positive value and that with the largest negative value in the stored pitch error compensation data for each axis. Four compensation positions can be set to 0 at a time. In this case, compensation is not performed.
5. To add the straightness compensation function, the stored pitch error compensation is needed.
   In this case, the number of compensation positions of each axis between the compensation position with the largest positive value and that with the largest negative value in the stored pitch error compensation data must be equal to or less than 1536.
6. Straightness compensation data is superposed on stored pitch error compensation data and output. Straightness compensation is performed at pitch error compensation intervals.
7. Straightness compensation does not allow the moving axis to be used as a compensation axis. To implement such compensation, use inclinaiton compensation (see Subsection "Inclination Compensation").

1.3.5 Interpolated Straightness Compensation

Overview

The interpolated straightness compensation function has the progress of the following two features as compared to the straightness compensation function. Therefore, more precise compensation can be realized.

- Straightness compensation at 128 points (The amount of compensation points increases.)
- Interpolated straightness compensation (Compensation data is output by dividing in between the pitch error compensation position.)

Table 1.3.5 (a) Functional comparison

	Straightness compensation	Interpolated straightness compensation
Amount of compensation points	4 points	Maximum 128 points
Interval of compensation	Interval of pitch error compensation position	Least command increment (*1)
Setting of compensation data	Parameter	PITCH ERROR COMPENSATION screen
Range of compensation data	- 32767 to 32767	-127 to 127 (*2)

(*1) Compensation data is output by dividing in between the pitch error compensation position on the moving axis.
(*2) Actual compensation data is the value that is multiplied by the magnification parameters (Nos. 13391 to 13396) to a set value.

> **NOTE**
> 1 To use this function, the stored pitch error compensation is required.
> 2 For the amount of compensation points of the moving axis, the interpolated straightness compensation function becomes the same as the stored pitch error compensation function. Therefore, the amount of pitch error compensation points of the moving axis must be within 128 points.

Explanation

- Specifying the compensation point number of the moving axis

1. Set the smallest compensation point number of the moving axis to parameter Nos. 13381 to 13386.
2. The following data are calculated automatically according to the stored pitch error compensation point number.
   - The largest compensation point number of the moving axis
   - The compensation point number for the reference position of the moving axis

Relationship of the stored pitch error compensation point number and the interpolated straightness compensation point number of the moving axis is in Fig. 1.3.5 (a)

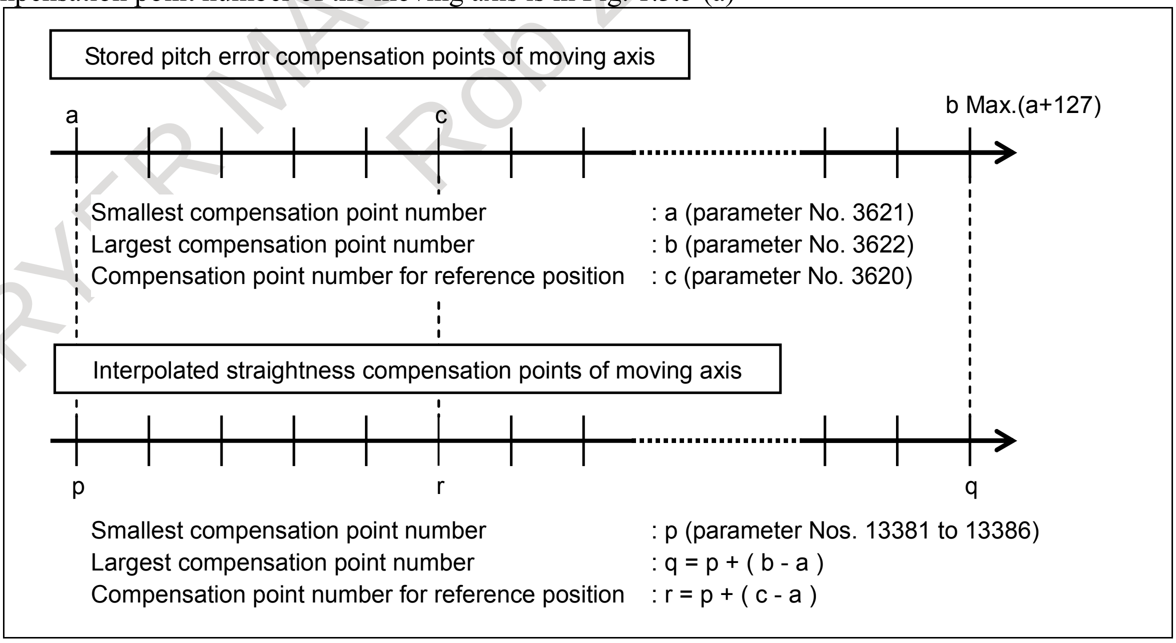


Fig. 1.3.5 (a) Relationship of the compensation point number

NOTE

In the interpolated straightness compensation function and the stored pitch error compensation function, the following data are the same value.

- Amount of compensation points
- Interval of compensation positions of the moving axis

- Setting of compensation data

Compensation data is set on the "PITCH ERROR COMPENSATION" screen.
Set it corresponding to the compensation point number of the moving axis. It is set in the range of -127 to 127 in the increase or decrease value from the previous compensation point number. And it is multiplied by the magnification parameters (Nos. 13391 to 13396).

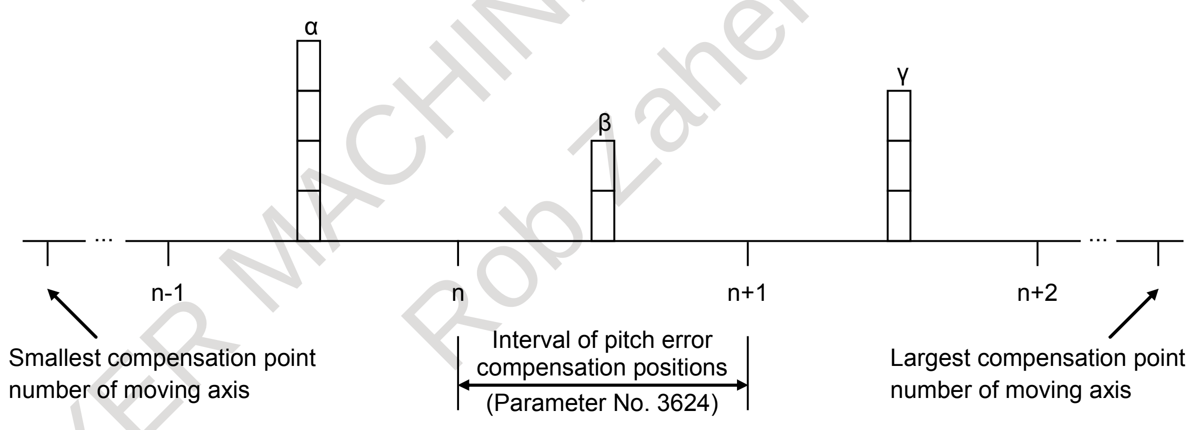
In addition to input with MDI unit on the screen, it is possible with the following ways.

- Input with G10 L50
- Input/output with the I/O unit interface
- Input from the PMC window (function code: 18)

In these methods, the compensation data can be input/output in parameter format in the same way as with the stored pitch error compensation data. In the interpolated straightness compensation function, parameter number is a value by adding 20000 to the compensation point number.
For more information, please refer to the description of each function.

- Compensation system

When the compensation data is set as shown in Fig. 1.3.5 (b), it is output by dividing in between the compensation position as shown in Fig. 1.3.5 (c). Therefore, the compensation axis becomes more smooth motion.



Compensation point number	...	n-1	n	n+1	n+2	...
Compensation data	...	...	α(4)	β(2)	γ(3)	...

Fig. 1.3.5 (b) Setting of compensation data

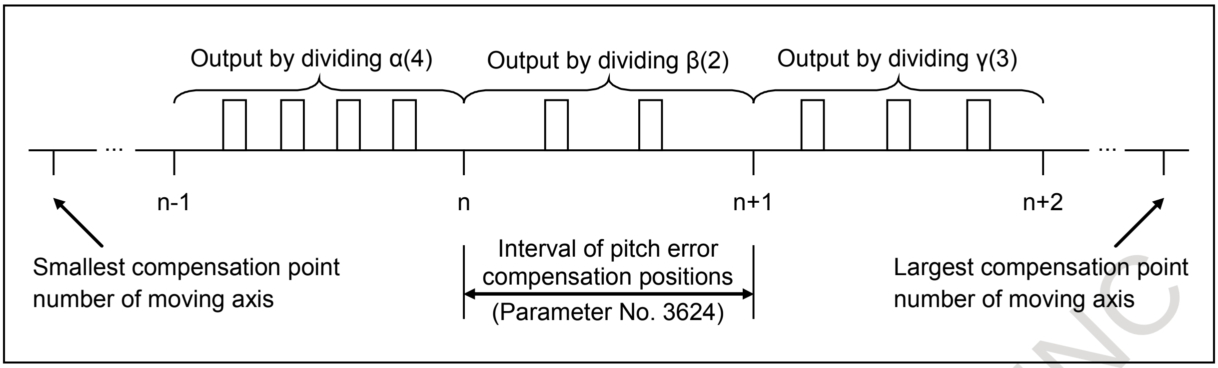


Fig. 1.3.5 (c) Interpolation system for interpolated straightness compensation

- Position display in interpolated straightness compensation

The interpolated straightness compensation amount is not reflected in position display.
The value in which interpolated straightness compensation amounts and other compensation amounts are integrated from power-on is displayed in diagnosis data No. 361.

- Setting of parameter

1. Set the parameter for stored pitch error compensation of the moving axis.
   - Compensation point number for reference position : Parameter No. 3620
   - Smallest compensation point number : Parameter No. 3621
   - Largest compensation point number : Parameter No. 3622
2. Set the parameter for interpolated straightness compensation.
   - Smallest compensation point number of moving axis : Parameter Nos. 13381 to 13386
   - Axis number of moving axis : Parameter Nos. 5711 to 5716
   - Axis number of compensation axis : Parameter Nos. 5721 to 5726
   - Magnification for compensation data : Parameter Nos. 13391 to 13396

Parameter setting examples

For the combination of the moving axis and the compensation axis, it can be set as shown in Table 1.3.5 (b) to Table 1.3.5 (e).

Table 1.3.5 (b) A single compensation axis can be set for a single moving axis.

Setting of moving axis		Setting of compensation axis		Effective magnification
Parameter No.	Setting	Parameter No.	Setting	
5711	1	5721	2	Value set in parameter No. 13391
5712	3	5722	4	Value set in parameter No. 13392
5713	5	5723	6	Value set in parameter No. 13393
5714	7	5724	8	Value set in parameter No. 13394
5715	9	5725	10	Value set in parameter No. 13395
5716	11	5726	12	Value set in parameter No. 13396

The 1st set : The 2nd axis is compensated according to the movement of the 1st axis.
The 2nd set : The 4th axis is compensated according to the movement of the 3rd axis.
The 3rd set : The 6th axis is compensated according to the movement of the 5th axis.
The 4th set : The 8th axis is compensated according to the movement of the 7th axis.
The 5th set : The 10th axis is compensated according to the movement of the 9th axis.
The 6th set : The 12th axis is compensated according to the movement of the 11th axis.

Table 1.3.5 (c) Two or more compensation axes can be set for a single moving axis.

Setting of moving axis		Setting of compensation axis		Effective magnification
Parameter No.	Setting	Parameter No.	Setting	
5711	1	5721	2	Value set in parameter No. 13391
5712	1	5722	3	Value set in parameter No. 13392
5713	1	5723	4	Value set in parameter No. 13393
5714	1	5724	5	Value set in parameter No. 13394

Setting of moving axis		Setting of compensation axis		Effective magnification
Parameter No.	Setting	Parameter No.	Setting	
5715	1	5725	6	Value set in parameter No. 13395
5716	1	5726	7	Value set in parameter No. 13396

The 1st set : The 2nd axis is compensated according to the movement of the 1st axis.
The 2nd set : The 3rd axis is compensated according to the movement of the 1st axis.
The 3rd set : The 4th axis is compensated according to the movement of the 1st axis.
The 4th set : The 5th axis is compensated according to the movement of the 1st axis.
The 5th set : The 6th axis is compensated according to the movement of the 1st axis.
The 6th set : The 7th axis is compensated according to the movement of the 1st axis.

Table 1.3.5 (d) A compensation axis can be set as a moving axis.

Setting of moving axis		Setting of compensation axis		Effective magnification
Parameter No.	Setting	Parameter No.	Setting	
5711	1	5721	2	Value set in parameter No. 13391
5712	2	5722	3	Value set in parameter No. 13392
5713	3	5723	4	Value set in parameter No. 13393
5714	4	5724	5	Value set in parameter No. 13394
5715	5	5725	6	Value set in parameter No. 13395
5716	6	5726	7	Value set in parameter No. 13396

The 1st set : The 2nd axis is compensated according to the movement of the 1st axis.
The 2nd set : The 3rd axis is compensated according to the movement of the 2nd axis.
The 3rd set : The 4th axis is compensated according to the movement of the 3rd axis.
The 4th set : The 5th axis is compensated according to the movement of the 4th axis.
The 5th set : The 6th axis is compensated according to the movement of the 5th axis.
The 6th set : The 7th axis is compensated according to the movement of the 6th axis.

NOTE
The movement by the compensation in one set, does not become the subject to compensation on the other set.

Table 1.3.5 (e) Two or more moving axes can be set for a single compensation axis.

Setting of moving axis		Setting of compensation axis		Effective magnification
Parameter No.	Setting	Parameter No.	Setting	
5711	1	5721	7	Value set in parameter No. 13391
5712	2	5722	7	Value set in parameter No. 13392
5713	3	5723	7	Value set in parameter No. 13393
5714	4	5724	7	Value set in parameter No. 13394
5715	5	5725	7	Value set in parameter No. 13395
5716	6	5726	7	Value set in parameter No. 13396

The 1st set : The 7th axis is compensated according to the movement of the 1st axis.
The 2nd set : The 7th axis is compensated according to the movement of the 2nd axis.
The 3rd set : The 7th axis is compensated according to the movement of the 3rd axis.
The 4th set : The 7th axis is compensated according to the movement of the 4th axis.
The 5th set : The 7th axis is compensated according to the movement of the 5th axis.
The 6th set : The 7th axis is compensated according to the movement of the 6th axis.

NOTE
When a compensation axis has two or more moving axes, it is compensated by the sum of the compensation data for each moving axis.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3605						IPCx		

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#2 IPCx Interpolated straightness compensation function is:
0: Not used.
1: Used.
Specify the value of this parameter for the moving axis.

	#7	#6	#5	#4	#3	#2	#1	#0
5700						SM2		

[Input type] Parameter input
[Data type] Bit path

#2 SM2 In the straightness compensation function, magnification parameters (parameters Nos. 13391 to 13396) are treated as follows:
0: When more than one moving axis is set with the same number, the setting of the magnification parameter for the moving axis set first is used.
1: When more than one moving axis is set with the same number, the setting of the magnification parameter for each axis is used.

5711	Straightness compensation : Axis number of moving axis 1
5712	Straightness compensation : Axis number of moving axis 2
5713	Straightness compensation : Axis number of moving axis 3
5714	Straightness compensation : Axis number of moving axis 4
5715	Straightness compensation : Axis number of moving axis 5
5716	Straightness compensation : Axis number of moving axis 6

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes
Set the axis numbers of moving axes for straightness compensation.
When 0 is set, compensation is not performed.

5721	Straightness compensation : Axis number of compensation axis 1 for moving axis 1
5722	Straightness compensation : Axis number of compensation axis 2 for moving axis 2
5723	Straightness compensation : Axis number of compensation axis 3 for moving axis 3

5724	Straightness compensation : Axis number of compensation axis 4 for moving axis 4
5725	Straightness compensation : Axis number of compensation axis 5 for moving axis 5
5726	Straightness compensation : Axis number of compensation axis 6 for moving axis 6

> **NOTE**
> When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

13381	Number of the straightness compensation point at the extremely negative position of moving axis 1
13382	Number of the straightness compensation point at the extremely negative position of moving axis 2
13383	Number of the straightness compensation point at the extremely negative position of moving axis 3
13384	Number of the straightness compensation point at the extremely negative position of moving axis 4
13385	Number of the straightness compensation point at the extremely negative position of moving axis 5
13386	Number of the straightness compensation point at the extremely negative position of moving axis 6

> **NOTE**
> When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 6000 to 6767

Set the number of the straightness compensation point at the extremely negative position for each moving axis.
When the value set in this parameter is out of the valid data range, an alarm is issued and compensation cannot be performed.

13391	Magnification for straightness compensation for moving axis 1
13392	Magnification for straightness compensation for moving axis 2
13393	Magnification for straightness compensation for moving axis 3
13394	Magnification for straightness compensation for moving axis 4
13395	Magnification for straightness compensation for moving axis 5
13396	Magnification for straightness compensation for moving axis 6

> **NOTE**
> When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 100

Set the magnification for straightness compensation for each moving axis.

When the magnification is set to 1, the unit of compensation data is the same as the detection unit. When the magnification is set to 0, straightness compensation is not performed.

Alarm and message

Number	Message	Description
SV1100	S-COMP.VALUE OVERFLOW	The straightness compensation has exceeded the maximum of 32767.
PW1103	ILLEGAL PARAMETER (S-COMP.128)	The setting of a parameter for straightness compensation at 128 points or the setting of compensation data is not correct.
PW5046	ILLEGAL PARAMETER (S-COMP)	The setting of a parameter related to straightness compensation contains an error. Possible causes include: - A non-existent axis number is set in a moving or compensation axis parameter. - The straightness compensation point numbers do not have correct magnitude relationships. - No straightness compensation point is found between the furthest pitch error compensation point in the negative region and that in the positive region. - The compensation per compensation point is either too large or too small.

Caution

⚠ **CAUTION**

1. To use this function, stored pitch error compensation is required.
2. The number of compensation points located between the furthest compensation point in the negative region and that in the positive region on each axis of stored pitch error compensation must not exceed 128.
3. The compensation point interval is the same as that of stored pitch error compensation (parameter No. 3624).
4. The compensation magnification can be set separately from that for stored pitch error compensation.
5. Straightness compensation is superposed with the data for stored pitch error compensation before being output.
6. If the motion value is high, multiple compensation pulses may be output at a time depending on the straightness compensation.
7. After setting parameters for straightness compensation, turn off the power to the NC and then back ON for the settings to take effect.
8. Interpolated straightness compensation cannot be used at the same time as conventional straightness compensation for a single moving axis. Interpolated straightness compensation can, however, be used together with conventional straightness compensation for different moving axes.

1.3.6 Interpolated Straightness Compensation 3072 Points

Overview

By adding this function to the interpolated straightness compensation, the number of compensation points which can be used is expanded to 3072. The number of points which can be used for one pair of interpolated straightness compensation is also expanded to 1536. As a result, higher accurate machining can be realized for a machine tool with a long stroke that requires straightness compensation, by more exact interpolated straightness compensation.

Explanation

Table 1.3.6 lists the number of compensation points which can be used for interpolated straightness compensation.

Table 1.3.6

	Interpolated straightness compensation	When interpolated straightness compensation 3072 points are added to interpolated straightness compensation
Points which can be set	768 points (used for all 6 pairs)	3072 points (used for all 6 pairs)
Points which can be set for one pair	128 points	1536 points
Setting screen	Set the points in Nos. 6000 to 6767 in the pitch error setting screen.	Set the points in Nos. 6000 to 9071 in the pitch error setting screen.

Caution

⚠ **CAUTION**

1 To use this function, the stored pitch error compensation and interpolated straightness compensation is required.
2 The compensation point interval is the same as that of stored pitch error compensation (parameter No. 3624).
3 The method for using this function is the same as for using interpolated straightness compensation. The related parameters are also the same as for interpolated straightness compensation. The valid data range of parameter Nos. 13381 to 13386 (number of the straightness compensation point at the extremely negative position) is changed to 6000 to 9071, however.
4 When this function is used, the number of compensation points located between the furthest compensation point in the negative region and that in the positive region on each axis of stored pitch error compensation must not exceed 1536.
5 Set parameters so that the total number of compensation points for moving axes used for six pairs of interpolated straightness compensation does not exceed 3072.

1.3.7 Inclination Compensation

Overview

By compensating for those errors in tools such as feed screws that depend on the position of the machine system in detection units, machining precision can be improved and mechanical life can be prolonged. Compensation is performed along an approximate straight line formed with a parameter-specified compensation point and a compensation amount related to it.

Explanation

Three approximate straight lines are formed with four parameter-specified compensation points and compensation amounts related to the respective compensation points. Inclinaiton compensation is carried out along these approximate straight lines at pitch error compensation intervals. The inclinaiton compensation amount is added to the pitch error compensation amount.

- Relation between pitch error compensation points and inclination compensation

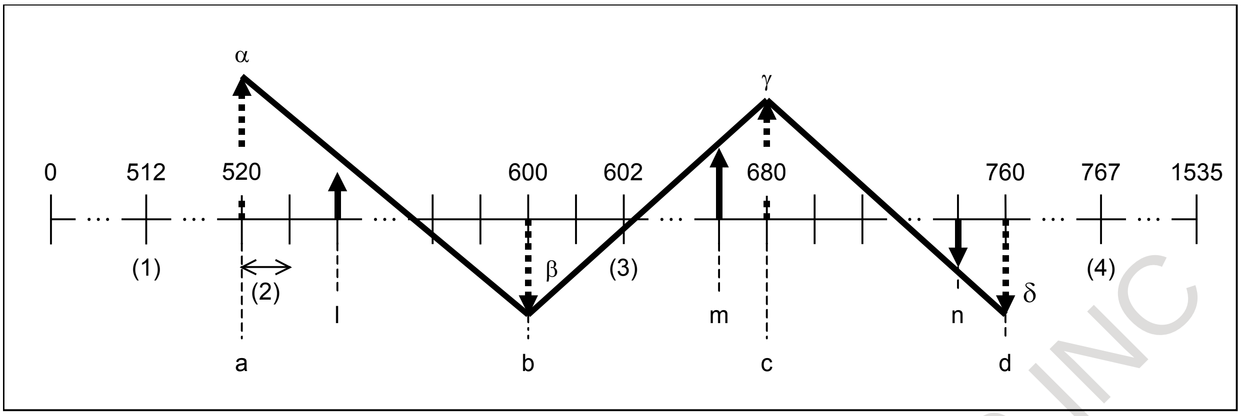


Fig. 1.3.7

To perform inclinaiton compensation, stored pitch error compensation must be set for the axis subject to compensation.

(1) Number of the most distant pitch error compensation point on the - side (parameter No.3621)
(2) Pitch error compensation point interval (parameter No. 3624)
(3) Number of the pitch error compensation point of the reference position (parameter No. 3620)
(4) Number of the most distant pitch error compensation point on the + side (parameter No. 3622)

InFig. 1.3.7, (1), (3) and (4) are 512, 602 and 767, respectively.

Inclinaiton compensation parameters must be set.

a,b,c,d : Compensation point numbers. (parameters Nos. 5861 to 5864)

α,β,γ,δ : Compensation amounts at compensation points a, b, c, and d (parameters Nos. 5871 to 5874)

In Fig. 1.3.4 (d)Fig. 1.3.7, a, b, c and d are 520, 600, 680, and 760, respectively.

Unlike stored pitch error compensation, whose amount is set up for an individual compensation point, an amount of inclinaiton compensation is calculated for individual compensation points by setting up four typical points and compensation amounts for them.

Example:

In above figure, the compensation amount at the individual compensation point is as follows

At compensation point l between compensation point a and point b:

$\alpha + (\beta-\alpha) / (b-a) \times (l-a)$

At compensation point m between compensation point b and point c:

$\beta + (\gamma-\beta) / (c-b) \times (m-b)$

At compensation point n between compensation point c and point d:

$\gamma + (\delta-\gamma) / (d-c) \times (n-c)$

Parameter

5861	**Inclination compensation : Compensation point number a for each axis**
5862	**Inclination compensation : Compensation point number b for each axis**
5863	**Inclination compensation : Compensation point number c for each axis**
5864	**Inclination compensation : Compensation point number d for each axis**

NOTE

When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1535
These parameters set the compensation points for inclination compensation. The points are set for the compensation point numbers for stored pitch error compensation.

5871	Inclination compensation : Compensation α at compensation point number a for each axis
5872	Inclination compensation : Compensation β at compensation point number b for each axis
5873	Inclination compensation : Compensation γ at compensation point number c for each axis
5874	Inclination compensation : Compensation δ at compensation point number d for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -32767 to 32767
Each of these parameters sets a compensation value for each axis compensation point.

Alarm and message

Number	Message	Description
PW1102	ILLEGAL PARAMETER (I-COMP.)	The parameter for setting slope compensation is incorrect. This alarm occurs in the following cases: • When the number of pitch error compensation points on the axis on which slope compensation is executed exceeds 1536 between the most negative side and most positive side • When the size relationship between the slope compensation point Nos. is incorrect • When the slope compensation point is not located between the most negative side and most positive side of pitch error compensation • When the compensation per compensation point is too small or too great.

Note

NOTE

1 To use this function, Stored pitch error compensation is required. The number of compensation points located between the most distant compensation point on the + side of each axis and the most distant compensation point on the + side in stored pitch error compensation must not exceed 1536.
2 Inclinaiton compensation is enabled after the reference position is established on the compensation axis.
3 When setting parameters Nos. 5861 to 5864 (compensation point numbers a to d for individual axes), turn off the power to the NC and then back ON for the settings to take effect.
4 During automatic operation, it is possible to overwrite parameters Nos. 5871 to 5874, but make sure that all axes are stopped beforehand. If any of parameters Nos. 5871 to 5874 (compensation amounts at compensation points a to d for individual axes) is changed, the compensation amount determined from the compensation amount after the change is output after the point at which to output the compensation amount for the next inclinaiton compensation is passed.

NOTE

5 Parameters must satisfy the following conditions:
   - The difference of compensation at a compensation point must be within the range -127 to 127. The difference of compensation at a compensation point between compensation point a and point b is $(\beta-\alpha)$ / (b-a).
   - Compensation points must satisfy the following relationships: $a \le b \le c \le d$.
   - Compensation points must be located between the most distant compensation point in stored pitch error compensation on the - side of each axis and the most distant compensation point on the + side. If all four points are equal to 0, compensation is not performed.

6 Inclinaiton compensation is superimposed on the stored pitch error compensation data.

7 This function is applied to both linear and rotation axes.

8 The compensation amount at the reference position is output based on parameter settings. The first compensation pulse is output when the compensation point is reached.

Warning

⚠ **WARNING**

If any of parameters Nos. 5871 to 5874 (compensation amounts at compensation points a to d for individual axes) is changed, very large compensation may be output depending on the setting. Great care should be taken.

1.3.8 Bi-directional Pitch Error Compensation

Overview

In bi-directional pitch error compensation, different pitch error compensation amounts can be set for travel in the positive direction and that in the negative direction, so that pitch error compensation can be performed differently in the two directions, in contrast to stored pitch error compensation, which does not distinguish between the directions of travel. In addition, when the direction of travel is reversed, the compensation amount is automatically calculated from the pitch error compensation data to perform compensation in the same way as in backlash compensation. This reduces the difference between the paths in the positive and negative directions.

Explanation

- Setting data

1. Setting parameters

   Set the following parameters for each axis. For detail of parameter setting, refer to the subsection of "Stored Pitch Error Compensation".

Table 1.3.8 (a)

Parameter number	Description
3605#0	Bidirectional pitch error compensation, 1: Enabled / 0: Disabled
3620	Number of the pitch error compensation point of the reference position
3621	Number of the most distant pitch error compensation point on the - side for travel in the positive direction
3622	Number of the most distant pitch error compensation point on the + side for travel in the positive direction
3623	Pitch error compensation magnification
3624	Pitch error compensation point interval
3625	For a rotary axis, amount of travel per rotation in pitch error compensation

Parameter number	Description
3626	Number of the most distant pitch error compensation point on the - side for travel in the negative direction
3627	Pitch error compensation amount (absolute value) at the reference position when the machine moves to the reference position in the direction opposite to that of a reference position return

2. Pitch error compensation data
   The compensation point numbers for bi-directional pitch error compensation can be from 0 to 1535 and from 3000 to 4535. This data may be used for both the positive and negative directions. Note, however, that the set of compensation data for a given axis cannot extend over 1535 and 3000.

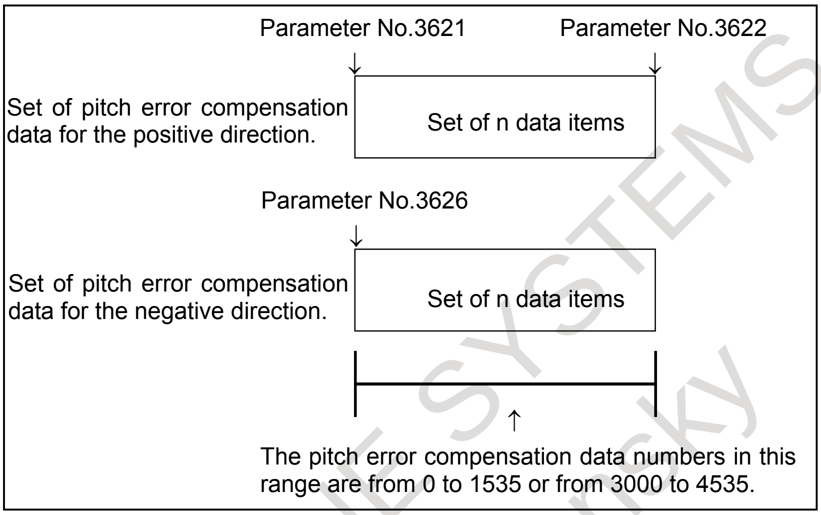


Fig. 1.3.8 (a)

- Data setting example

If the direction of a manual reference position return is positive on an axis (linear axis) having the pitch error amounts shown in the figure below (Fig. 1.3.8 (b)), set the data given in the table below (Table 1.3.8 (b)).

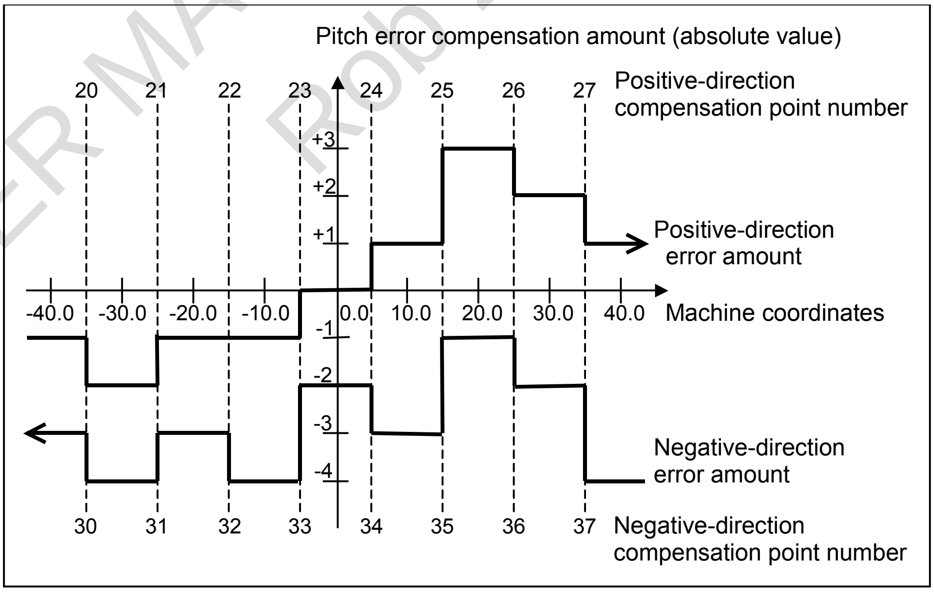


Fig. 1.3.8 (b)

Table 1.3.8 (b) Positive-direction pitch error data

Compensation point number	20	21	22	23	24	25	26	27
Compensation amount to be set	-1	+1	0	+1	+1	+2	-1	-1

As pitch error data, always set incremental values as viewed in the negative direction (direction toward the left in Fig. 1.3.8 (b)).

Table 1.3.8 (c) Negative-direction pitch error data

Compensation point number	30	31	32	33	34	35	36	37
Compensation amount to be set	-1	+1	-1	+2	-1	+2	-1	-2

Set negative-direction pitch error data for all the points for which positive-direction pitch error data has been set.
As negative-direction pitch error data, always set incremental values as viewed in the positive-direction.

Table 1.3.8 (d)

Parameter number	Setting	Description
3605#0	1	Bi-directional pitch error compensation, 1: Enabled / 0: Disabled
3620	23	Number of the pitch error compensation point for the reference position
3621	20	Number of the most distant pitch error compensation point on the - side for travel in the positive direction
3622	27	Number of the most distant pitch error compensation point on the + side for travel in the positive direction
3623	1	Pitch error compensation magnification
3624	10.0	Pitch error compensation point interval
3625	-	For a rotary axis, amount of travel per rotation in pitch error compensation
3626	30	Number of the most distant pitch error compensation point on the - side for travel in the negative direction
3627	-2	Pitch error compensation amount (absolute value) at the reference position when the machine moves to the reference position in the direction opposite to that of the reference position return

This example assumes that the direction of a manual reference position return is positive. For parameter No. 3627, therefore, set -2, which is the pitch error compensation amount (absolute value) at the reference position when the machine moves to the reference position in the negative direction.

- Compensation example

If, in the setting example given in the previous section, the machine moves

0.0 to 40.0,
40.0 to -40.0, and
-40.0 to 0.0

for a manual reference position return, pitch error compensation pulses are output as follows:

Machine coordinate	0.0	5.0	15.0	25.0	35.0	40.0
Compensation pulse	-	+1	+2	-1	-1	-5

Machine coordinate	35.0	25.0	15.0	5.0	-5.0	-15.0	-25.0	-35.0	-40.0
Compensation pulse	+2	+1	-2	+1	-2	+1	-1	+1	+2

Machine coordinate	-35.0	-25.0	-15.0	-5.0	0.0
Compensation pulse	-1	+1	0	+1	-

When the travel direction changes from positive to negative at the position of 40.0, the compensation for the reverse of the travel direction is output.

A pulse of -5 is the result of the following calculation:

-5 = (-4) - (+1)

(+1): Pitch error associated with the positive-direction absolute value at the position of 40.0

(-4): Pitch error associated with the negative-direction absolute value at the position of 40.0

Fig. 1.3.8 (c)

When the travel direction changes from negative to positive at the position of -40.0, the compensation for the reverse of the travel direction is output.

A pulse of +2 is the result of the following calculation:

+2 = (-1) - (-3)

(-3): Pitch error associated with the negative-direction absolute value at the position of -40.0

(-1): Pitch error associated with the positive-direction absolute value at the position of -40.0

Fig. 1.3.8 (d)

- Setting and displaying data

All pitch error the compensation data can be displayed and set on the conventional screen for the pitch error compensation data.

And those data can be input and output by the following methods.

- Input by MDI
- Input by G10
- Input and output by input/output device interface
- Input by PMC window (function code 18)

(It is not possible to input and output by the method other than the above methods.)

Compensation data for bi-directional pitch error compensation can be input/output in parameter format in the same way as stored pitch error compensation data. A pitch error compensation point number plus 20000 is the corresponding parameter number. (The format is the same as that for pitch error compensation data.)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3605								**BDPx**

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 BDPx Both-direction pitch error compensation is:

0: Not used.
1: Used.

3620 **Number of the pitch error compensation position for the reference position for each axis**

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1535
Set the number of the pitch error compensation position for the reference position for each axis.

3621 **Number of the pitch error compensation position at extremely negative position for each axis**

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1535
Set the number of the pitch error compensation position at the extremely negative position for each axis.

3622 **Number of the pitch error compensation position at extremely positive position for each axis**

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1535
Set the number of the pitch error compensation position at the extremely positive position for each axis.
This value must be larger than set value of parameter No.3620.

3623 **Magnification for pitch error compensation for each axis**

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 100
Set the magnification for pitch error compensation for each axis.
If the magnification is set to 1, the same unit as the detection unit is used for the compensation data.
If 0 is set, compensation is not performed.

3624	Interval between pitch error compensation positions for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] See the description below.

The pitch error compensation positions are arranged with equal spacing. The space between two adjacent positions is set for each axis. The minimum interval between pitch error compensation positions is limited and obtained from the following equation:
Minimum interval between pitch error compensation positions = maximum feedrate/7500
Unit : mm, inch, deg or mm/min, inch/min, deg/min
Example:
When the maximum feedrate is 15000 mm/min, the minimum interval between pitch error compensation positions is 2 mm.

3625	Travel distance per revolution in pitch error compensation of rotary axis type

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] See the description below.

If the pitch error compensation of rotary axis type is performed (bit 1 (ROSx) of parameter No. 1006 is set to 0 and bit 0 (ROTx) of parameter No. 1006 is set to 1), set the travel distance per revolution. The travel distance per revolution does not have to be 360 degrees, and a cycle of pitch error compensation of rotary axis type can be set.
However, the travel distance per revolution, compensation interval, and number of compensation points must satisfy the following condition:
(Travel distance per revolution) = (Compensation interval) × (Number of compensation points)
The compensation at each compensation point must be set so that the total compensation per revolution equals 0.

NOTE
If 0 is set, the travel distance per revolution becomes 360 degrees.

3626	Number of the both-direction pitch error compensation position at extremely negative position (for movement in the negative direction)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1535, 3000 to 4535

When using both-direction pitch error compensation, set the number of compensation point at the farthest end in the negative direction for a movement in the negative direction.

> **NOTE**
> 1 For a movement in the positive direction, set the compensation point number at the farthest end in the negative direction in parameter No. 3621.
> 2 A set of compensation data items for a single axis should not be set to lie astride 1535 and 3000.

3627	Pitch error compensation at reference position when a movement to the reference position is made from the direction opposite to the direction of reference position return

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -32768 to 32767

Set the absolute value of pitch error compensation at reference position when a movement to the reference position is made from the negative direction if the direction of reference position return (bit 5 (ZMI) of parameter No. 1006) is positive or from the positive direction if the direction of reference position return is negative.

Note

> **NOTE**
> 1 To use this function, Stored pitch error compensation is required.
> 2 This function is enabled after a manual reference position return or an automatic reference position return with the same sequence as that of a manual reference position return is performed. When an absolute position detector is used, however, the function is enabled after the power is turned on.
> 3 When the machine moves to the reference position in the reference position return direction, set the absolute value of the pitch error compensation pulse to 0.
> 4 When this function and backlash compensation are used at the same time, the pulse resulting from backlash compensation is superimposed on the compensation pulse when the travel direction is reversed.
> 5 When this function is used for a rotary axis, the sum of the pitch error compensation amounts per rotation about the rotary axis must be 0 for both the positive and negative directions.
> 6 The function cannot be used with the inclination compensation function.

1.3.9 Interpolation Type Pitch Error Compensation

Overview

In stored pitch error compensation, the pitch error compensation pulse at each pitch error compensation point is output in the interval between that point and the next compensation point, as shown in the figure below (Fig. 1.3.9 (a)).

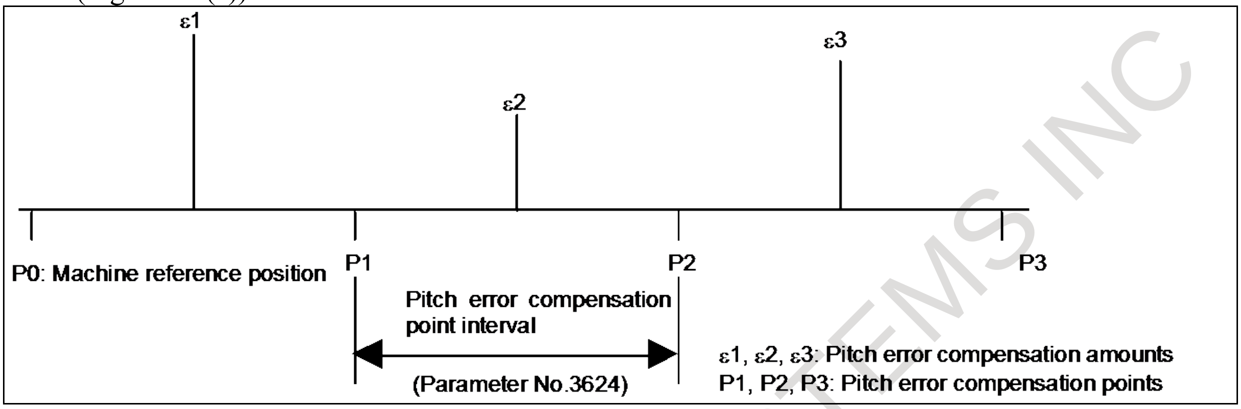


Fig. 1.3.9 (a) Stored pitch error compensation

In interpolation type pitch error compensation, the compensation amount at each error compensation point is divided into pulses in the interval between that point and the next point on the travel axis and output, as shown in the figure below. (Fig. 1.3.9 (b))

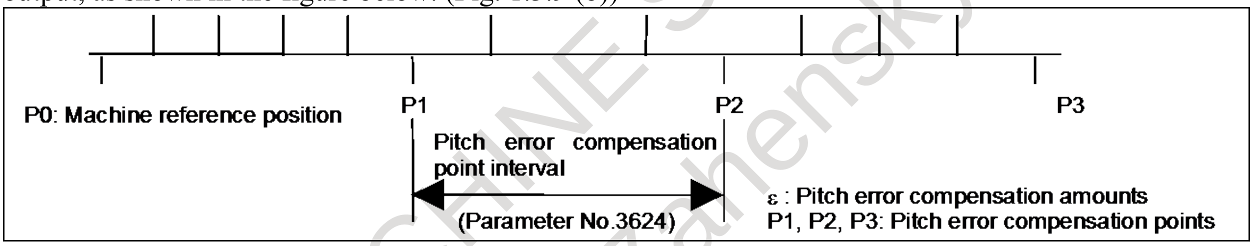


Fig. 1.3.9 (b) Interpolation Type Pitch Error Compensation Method

- Setting the parameters

When interpolation type pitch error compensation is used, the following parameters are assigned the same values as those in stored pitch error compensation.

- Number of the pitch error compensation point of the reference position on each axis:
  Parameter No.3620
- Number of the most distant pitch error compensation point on the - side of each axis:
  Parameter No.3621
- Number of the most distant pitch error compensation point on the + side of each axis:
  Parameter No.3622
- Pitch error compensation magnification for each axis: Parameter No.3623
- Pitch error compensation point interval on each axis: Parameter No.3624

- Minimum pitch error compensation point interval

Compensation pulses are divided at intervals of pitch error compensation point interval and output by interpolation type pitch error compensation,
Interpolation type pitch error compensation outputs compensation pulses smoother than stored pitch error compensation. But if the feedrate is high, multiple compensation pulses may be output at the same time.
The setting pitch error compensation point interval greater than minimum value calculated by the following formula had better be used in order to output smoother compensation pulses even if the feedrate is high.
Minimum pitch error compensation point interval (mm)= (Fmax/7500)×(Pmax+1)
Fmax : Maximum feedrate(mm/min)

Pmax : Maximum pitch error compensation amount((Maximum value of set amount of pitch error compensation of each compensation point)×(Pitch error compensation magnification))(Detection unit)
Example)
If the maximum feedrate is 15000 mm/min and the maximum pitch error compensation amount is equal to seven pulses, the minimum compensation interval is 16mm.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3605							IPPx	

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 IPPx Interpolation type pitch error compensation is:
0: Not used.
1: Used.

In interpolation type pitch error compensation, a compensation value at each point in each error completion point interval is divided for output of one pulse at equally spaced intervals.
If a high feedrate is used, multiple compensation pulse may be output at a time.
A minimum interval where multiple compensation pulses are not output at a time is determined by the following expression:
Minimum pitch error compensation point interval (mm)= (Fmax/7500) × (Pmax+1)
Fmax: Maximum feedrate(mm/min)
Pmax: Maximum pitch error compensation value((Maximum value of set amount of pitch error compensation of each compensation point)×(Pitch error compensation magnification))(Detection unit)
Example:
When the maximum feedrate is 15000 mm/min, and the maximum pitch error compensation value is 7 pulses, the minimum compensation point interval is 16mm.

> **NOTE**
> Interpolation type pitch error compensation cannot be used with spindle positioning.

Note

> **NOTE**
> 1 To use this function, Stored pitch error compensation is required.
> 2 This function is available in bi-directional pitch error compensation.

1.3.10 About Differences among Pitch Error Compensation, Straightness Compensation, and Inclinaiton Compensation (for Reference Purposes)

Overview

Any of pitch error compensation, straightness compensation, and inclinaiton compensation is applied to each compensation point based on the machine position at parameter-specified compensation intervals into which the machine stroke is divided.
Both inclinaiton compensation and straightness compensation use the same compensation intervals and compensation points as for pitch error compensation. However, they use their own compensation amounts defined for respective compensation functions.

Explanation

- Pitch error compensation

For pitch error compensation, a compensation amount is set up for each compensation point. The compensation amount is output at each compensation point.

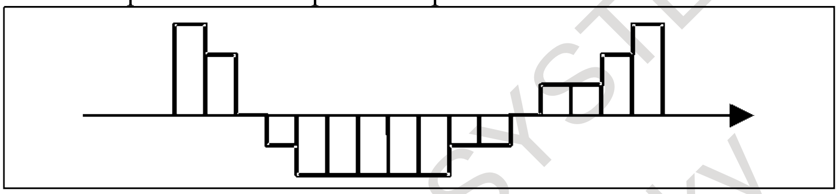


Fig. 1.3.10 (a)

- Bi-directional pitch error compensation

For bi-directional pitch error compensation, a compensation amount can be varied according to the axis movement direction.

- Interpolation type pitch error compensation

Interpolation type pitch error compensation outputs divided compensation pulses between compensation points, so smoother pitch error compensation can be realized.

- Inclinaiton compensation

In inclinaiton compensation, four typical pitch error compensation points (a, b, c, and d) are selected from pitch error compensation points and specified as inclinaiton compensation points, and compensation amounts are set up only for these four points; a compensation amount is not set up for every individual point.
For pitch error compensation points between inclinaiton compensation points, the NC calculates and outputs amounts that match inclinaiton compensation. Inclinaiton compensation can be applied if a pitch error has a constant gradient.

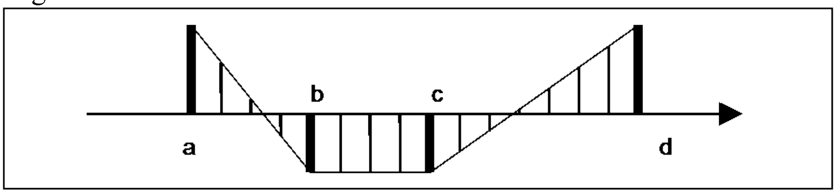


Fig. 1.3.10 (b)

- Straightness compensation

In straightness compensation, similarly to inclinaiton compensation, four typical pitch error compensation points (a, b, c, and d) are selected from pitch error compensation points and specified as straightness compensation points, and compensation amounts are set up only for these four points. For pitch error compensation points between straightness compensation points, the NC calculates and outputs amounts that match straightness compensation.

Straightness compensation largely differs from inclinaiton compensation in that the moving axis is not a compensation axis; inclinaiton compensation is applied directly to the moving axis. This relationship is specified by a parameter (for example, to apply compensation to the Y-axis as movement occurs along the X-axis).

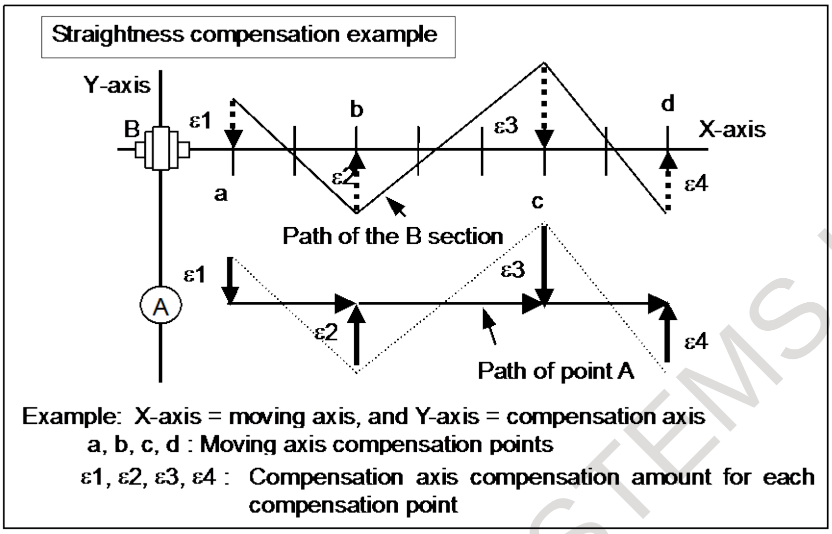


Fig. 1.3.10 (c)

1.3.11 Axis Name Display of Pitch Error Compensation

On the pitch error compensation screen, the axis name can be displayed to the left of the number of each compensation point used for pitch error compensation of each axis.
Whether to display the axis name can be selected using bit 5 (PAD) of parameter No. 11350.
When this function is enabled, the pitch error compensation screen is displayed as shown in Fig. 1.3.11 (a) or Fig. 1.3.11 (b).

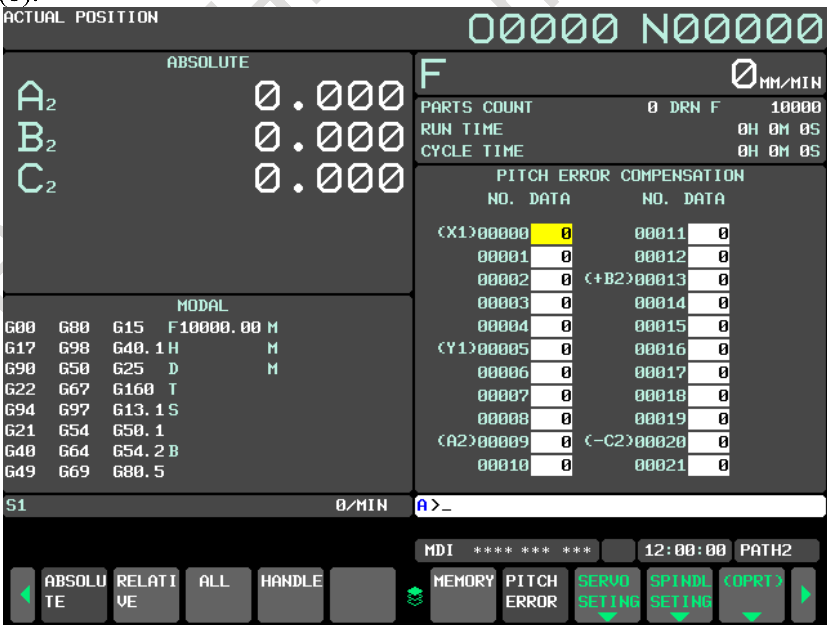


Fig. 1.3.11 (a) Pitch error compensation screen
(when the bi-directional pitch error compensation function is enabled and axis name extension is disabled)

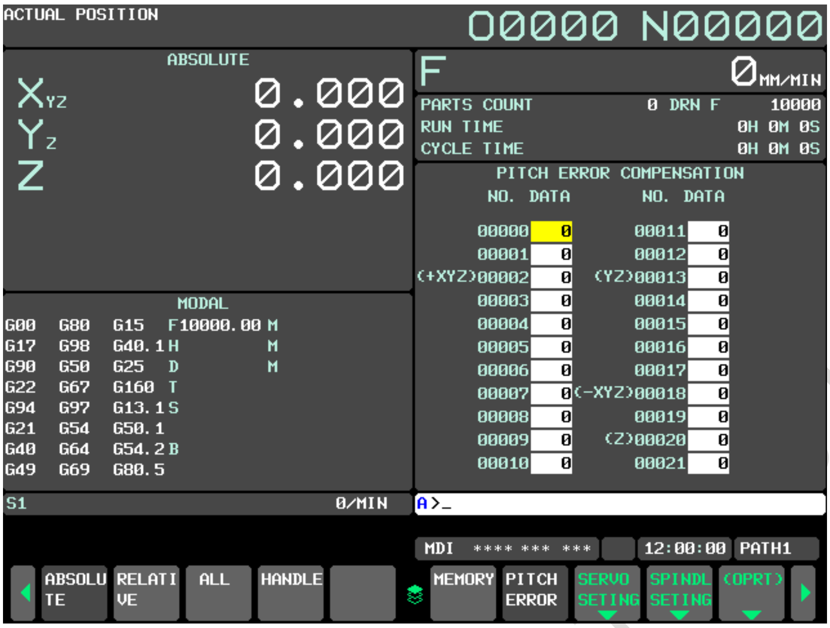


Fig. 1.3.11 (b) Pitch error compensation screen
(when the bi-directional pitch error compensation function
and axis name extension are enabled)

1.3.11.1 Setting of axis name display

The axis name is displayed to the left of the compensation point number set in parameter No. 3621, which sets the number of the pitch error compensation point at the extremely negative position. When the bi-directional pitch error compensation function is enabled, "(+ axis name)" is displayed to the left of the compensation point number set in parameter No. 3621, which sets the compensation point number for a movement in the positive direction, and "(- axis name)" is displayed to the left of the compensation point number set in parameter No. 3626, which sets the compensation point number for a movement in the negative direction.

> **NOTE**
> 1 When an invalid parameter is set for pitch error compensation, the relevant axis name is not displayed.
> 2 For a rotary axis, the axis name is displayed to the left of the number of the pitch error compensation point for the reference position set in parameter No. 3620.

1.3.11.2 Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11350			PAD					

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#5 PAD On the pitch error compensation screen, axis names are:
0: Not displayed.
1: Displayed.

1020	Program axis name for each axis

[Input type] Parameter input
[Data type] Byte axis
Set a program axis name for each axis.

> **NOTE**
> When this parameter is not set, the axis name displayed on the pitch error compensation screen is “(X)”.

3621	Number of the pitch error compensation position at extremely negative position for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
Set the number of the pitch error compensation position at the extremely negative position for each axis.

3626	Number of the both-direction pitch error compensation position at extremely negative position (for movement in the negative direction)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
When using both-direction pitch error compensation, set the number of compensation point at the farthest end in the negative direction for a movement in the negative direction.

3620	Number of the pitch error compensation position for the reference position for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
Set the number of the pitch error compensation position for the reference position for each axis.

1.3.12 Stored Pitch Error Compensation Total Value Input function

General

The stored pitch error compensation data can be input as a total value whose origin is the reference position.
In the conventional specification, it is necessary to calculate the difference between two consecutive compensation data. But the measured value that is a total value from the base point can be input directly when bit 0 (APE) of parameter No. 3602 is set to 1.

This function is effective to the following functions.
- Stored Pitch Error Compensation
- Bi-directional Pitch Error Compensation
- Interpolation Type Pitch Error Compensation
- Interpolation Type Straightness Compensation
- Spindle Command Synchronous Control Independent Pitch Error Compensation

NOTE
1 Stored Pitch Error Compensation is necessary to use this function.
2 If bit 0 (APE) of parameter No. 3602 is changed, the data of stored pitch error compensation is cleared automatically at next power on.

Explanation

If bit 0 (APE) of parameter No. 3602 is set to 1, pitch error compensation screen is enhanced as shown in the following Fig.1.3.12 (b), and inputting data becomes a total value from the base point.

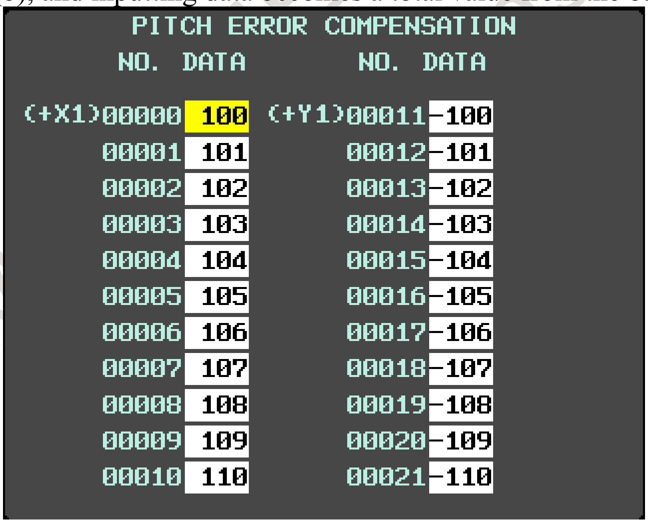


Fig.1.3.12 (a) Input screen by incremental value

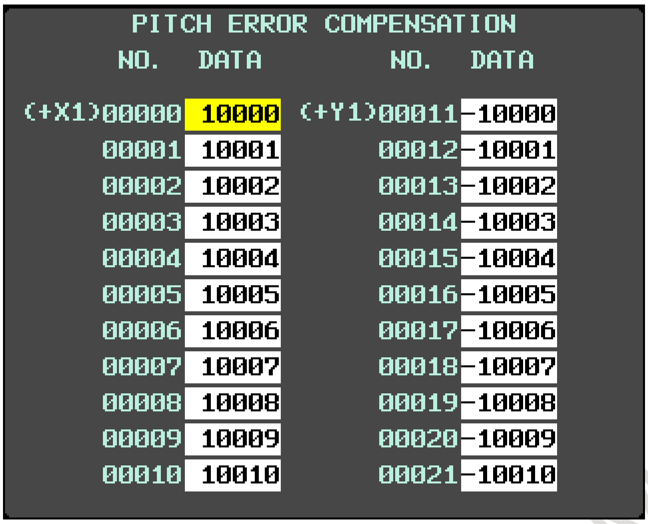


Fig.1.3.12 (b) Input screen by total value

The compensation data is input by a total value whose origin is the reference position. Please set 0 to the compensation data at the reference position.
The range of the value is –32768 to +32767. If the input value exceeds this range, the warning massage “DATA IS OUT OF RANGE” is displayed.
The difference between two consecutive data must be within –128 to +127. If the difference exceeds this range, the warning message “DIFFERENCE WITH NEAR IS RANGE OVER” is displayed and the compensation value is displayed in red color. When this warning is generated, if a program is executed in any path, alarm PS0527 “ILLEGAL DATA IN PITCH ERROR” occurs.

The compensation number that is checked the difference of compensation data is decided by considering the setting of the parameters shown in Table 1.3.12 (a).

Table 1.3.12 (a) Parameters that decides effective compensation number.

Parameter No.	Description
No.3601#1	Spindle command synchronous control independent pitch error compensation is not used (0) / used (1)
No.3602#0	The input type of stored pitch error compensation data is an incremental value (0) / a total value (1)
No.3605#0	Bi-directional pitch error compensation is not used (0) / used (1)
No.3605#2	Interpolation type straightness compensation is not used (0) / used (1)
No.3621	Number of the pitch error compensation point at extremely negative position.
No.3622	Number of the pitch error compensation point at extremely positive position.
No.3623	Pitch error compensation magnification.
No.3624	Pitch error compensation point interval.
No.3626	Number of the bi-directional pitch error compensation point at extremely negative position. (for travel in the negative direction)
No.3666	Number of the spindle command synchronous control independent pitch error compensation point at extremely negative position.
No.3671	Number of the spindle command synchronous control independent pitch error compensation point at extremely positive position.
No.3676	Number of the spindle command synchronous control independent bi-directional pitch error compensation point at extremely negative position. (for travel in the negative direction)
No.5711 to No.5716	Axis number of moving axis for straightness compensation.
No.5721 to No.5726	Axis number of compensation axis for straightness compensation.
No.13381 to No.13386	Number of the interpolation type straightness compensation point at extremely negative position.

Example1)

When bi-directional pitch error compensation and interpolation type straightness compensation are enabled, if parameters in Table 1.3.12 (b) are set, the compensation number that is checked the difference of compensation data is

No.100 to No.200 : The compensation number of stored pitch error compensation.
No.3100 to No.3200: The compensation number of bi-directional pitch error compensation.
No.6000 to No.6100: The compensation number of interpolation type straightness compensation.

Table 1.3.12 (b) Sample setting 1

Parameter No.	Setting value
No.3602#0	1
No.3605#0 (1st-axis)	1
No.3605#2 (1st-axis)	1
No.3621 (1st-axis)	100
No.3622 (1st-axis)	200
No.3623 (1st-axis)	1
No.3624 (1st-axis)	10.0
No.3626 (1st-axis)	3100
No.5711	1
No.5721	2
No.13381	6000

Example2)

When only stored pitch error compensation and bi-directional pitch error compensation are specified, pitch error compensation number 0 to 1535 and 3000 to 4535 are available. If parameters in Table 1.3.12 (c) are set, the compensation number that is checked the difference of compensation data is

No.100 to No.200 : The compensation number of stored pitch error compensation.
No.4500 to No.4600: The compensation number of bi-directional pitch error compensation.

Table 1.3.12 (c) Sample setting 2

Parameter No.	Setting value
No.3602#0	1
No.3605#0 (1st-axis)	1
No.3621 (1st-axis)	100
No.3622 (1st-axis)	200
No.3623 (1st-axis)	1
No.3624 (1st-axis)	10.0
No.3626 (1st-axis)	4500

As the No.4536 to No.4600 are not available, the data of these numbers are assumed to be 0. If the data set in No.4535 is larger than 128, the difference of compensation data between No.4535 and No.4536 is also over -128. This will be cause of warning “DIFFERENCE WITH NEAR IS RANGE OVER”.

The specification other than total value input is same as the conventional function. Please refer to the specification of the following functions.

- Stored Pitch Error Compensation
- Bi-directional Pitch Error Compensation
- Interpolation Type Pitch Error Compensation
- Interpolation Type Straightness Compensation
- Spindle Command Synchronous Control Independent Pitch Error Compensation

- Method of inputting and outputting compensation data on pitch error compensation screen

If the input type of stored pitch error compensation data is a total value (bit 0 (APE) of parameter No. 3602 is set to 1), the formats to output by the operation on pitch error compensation screen are the following.

The L1 data is added to the conventional format.

Example of stored pitch error compensation)

N10000 Q0 L1 P1000
N10001 Q0 L1 P995
N10002 Q0 L1 P990
N10003 Q0 L1 P995

Example of bi-directional pitch error compensation)

N20000 Q0 L1 P1000
N20001 Q0 L1 P995
N20002 Q0 L1 P990
N20003 Q0 L1 P995

This data can be input on the pitch error compensation screen when bit 0 (APE) of parameter No. 3602 is set to 1. If the data with L1 is input when bit 0 (APE) of parameter No. 3602 is set to 0, or the data without L1 is input when bit 0 (APE) of parameter No. 3602 is set to 1, alarm SR1300 “ILLEGAL ADDRESS” is caused.

- Input of pitch error compensation data by program

When bit 0 (APE) of parameter No. 3602 is set to 1, please execute the program by adding the L1 data to the conventional format.

Example of G10 command:

```
G10L50;
N10000 L1 R1000;
N10001 L1 R995;
N10002 L1 R990;
N10003 L1 R995;
G11;
```

If the program with L1 is executed when bit 0 (APE) of parameter No. 3602 is set to 0 or the program without L1 is executed when bit 0 (APE) of parameter No. 3602 is set to 0, alarm PS1300 “ILLEGAL ADDRESS” is caused.

NOTE

If the compensation data is changed when the input format is total value type(bit 0 (APE) of parameter No. 3602 is set to 1), power must be turned off before operation is continued.

- Example of total value input in case of bi-directional pitch error compensation

When the axis is linear axis and the direction of a manual reference position return is positive, set the data given in the Table 1.3.12 (d), if the pitch error amounts should be set in the Fig. 1.3.12 (c).

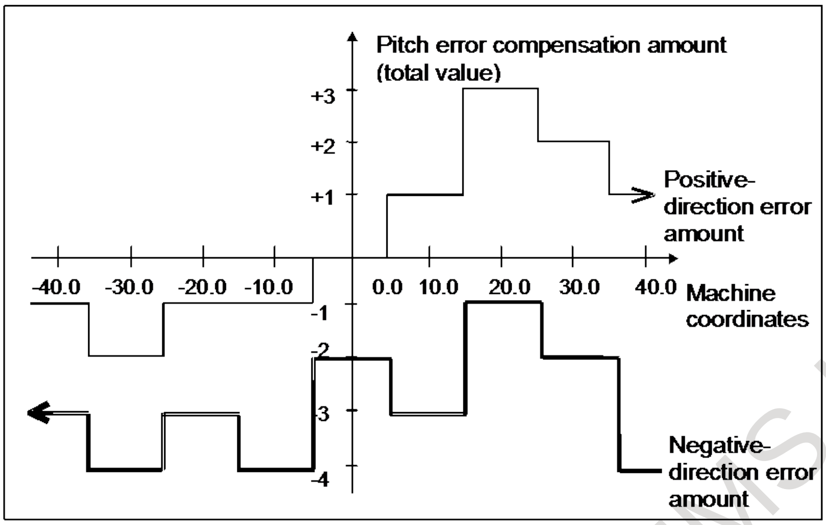


Fig. 1.3.12 (c) Example of bi-directional pitch error compensation

Table 1.3.12 (d) Parameter setting of bi-directional pitch error compensation

Parameter number	Setting of Input by Incremental value	Setting of Input by Total value	Description
3602#0	0	1	The input type of stored pitch error compensation data is, 0: An incremental value / 1: A total value
3605#0	1	1	Bi-directional Pitch Error Compensation, 0: Disabled / 1: Enabled
3620	23	24	Number of the pitch error compensation point for the reference position
3621	20	20	Number of the most distant pitch error compensation point on the - side for travel in the positive direction
3622	27	28	Number of the most distant pitch error compensation point on the + side for travel in the positive direction
3623	1	1	Pitch error compensation magnification
3624	10.000	10.000	Pitch error compensation point interval
3625	-	-	For a rotation axis, amount of travel per rotation in pitch error compensation
3626	30	30	Number of the most distant pitch error compensation point on the - side for travel in the negative direction
3627	-2	0	Pitch error compensation amount (total value) at the reference position when the machine moves to the reference position in the direction opposite to that of the reference position return

This example assumes that the direction of a manual reference position return is positive.
In case of Input by Incremental value:
The variation at the division of compensation areas is set.
As for parameter No. 3627, set -2, which is the pitch error compensation amount (total value) at the reference position when the machine moves in the negative direction.
In case of Input by Total value:
The value at the center of the compensation area is set. Therefore, one compensation point increases.
The parameter No. 3627 is not used.

Positive-direction pitch error data is as Table 1.3.12 (e).

Table 1.3.12 (e) Compensation value in positive direction

Compensation point number	Input by Incremental value	Input by Total value
20	-1	-1
21	+1	-2
22	0	-1
23	+1	-1
24	+1	0
25	+2	+1
26	-1	+3
27	-1	+2
28	-	+1

Negative-direction pitch error data is as Table 1.3.12 (f).

Table 1.3.12 (f) Compensation value in negative direction

Compensation point number	Input by Incremental value	Input by Total value
30	-1	-3
31	+1	-4
32	-1	-3
33	+2	-4
34	-1	-2
35	+2	-3
36	-1	-1
37	-2	-2
38	-	-4

Example of total value input in case of rotary axis

- Amount of movement per rotation: 360°
- Interval between pitch error compensation positions: 45°
- Number of the compensation position of the reference position: 60

In the above case, the number of the most distance compensation position on the - side is equal to the number of the compensation position of the reference position + 1 = 60 + 1 = 61 for a rotary axis.
The number of the farthest compensation position in the positive direction is as follows:
Number of the compensation position of the reference position + (Move amount per rotation/Interval between the compensation positions)= 60 + 360/45= 68
The correspondence between the machine coordinate and the compensation position number is as Fig. 1.3.12 (d)

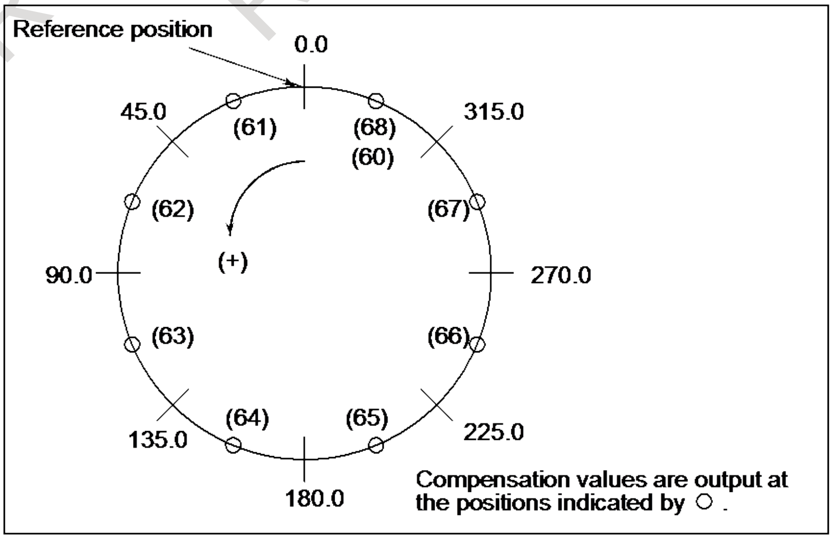


Fig. 1.3.12 (d) Example of rotary axis pitch error compensation

Therefore, set the parameters as Table 1.3.12 (g).

The parameter setting for the rotary axis in case of total value input is same as that of incremental value input.

Table 1.3.12 (g) Parameter setting of rotary axis pitch error compensation

Parameter number	Setting of Input by Incremental value	Setting of Input by Total value	Description
3602#0	0	1	The input type of stored pitch error compensation data is, 0: An incremental value / 1: A total value
3620	60	60	Number of the pitch error compensation point for the reference position
3621	61	61	Number of the most distant pitch error compensation point on the - side for travel in the positive direction
3622	68	68	Number of the most distant pitch error compensation point on the + side for travel in the positive direction
3623	1	1	Pitch error compensation magnification
3624	45.000	45.000	Pitch error compensation point interval
3625	360.000	360.000	For a rotation axis, amount of travel per rotation in pitch error compensation

When incremental value input is enabled, if the sum of the compensation values for positions 61 to 68 is not 0, pitch error compensation values are accumulated for each rotation, causing positional deviation. When total value input is enabled, set the value 0 in compensation positions 60 and 68.

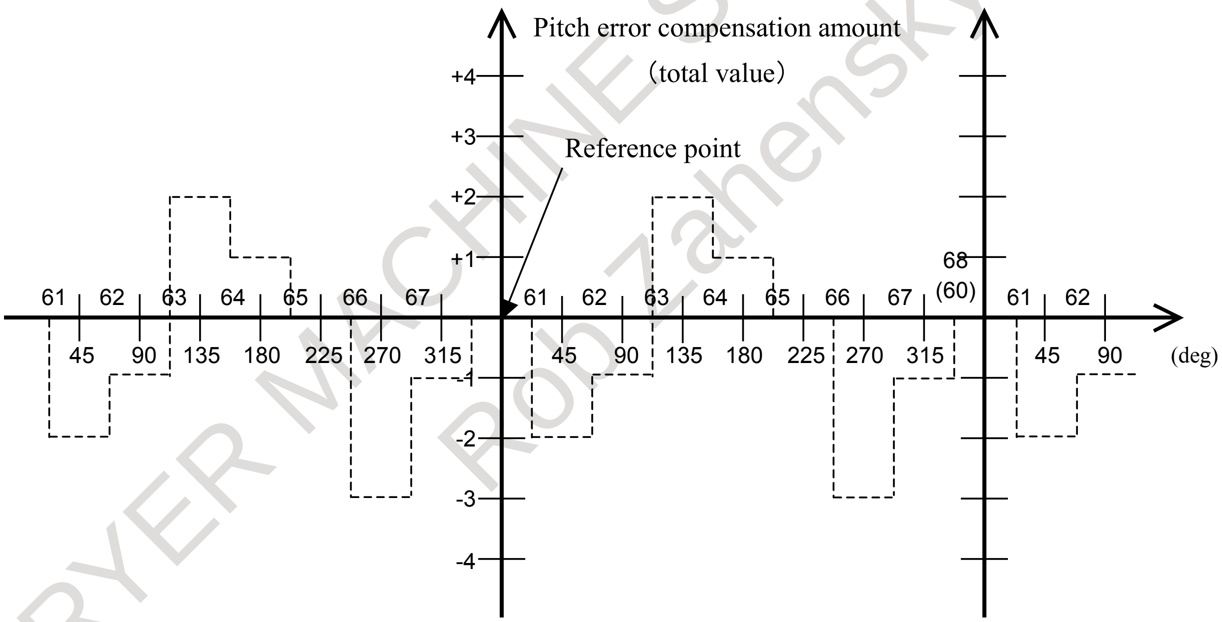


Fig. 1.3.12 (e) Compensation value of rotary axis expressed with linear axis

Table 1.3.12 (h) Compensation value of rotary axis

Compensation point number	(deg)	Input by Incremental value	Input by Total value
60	0	+1	0
61	45	-2	-2
62	90	+1	-1
63	135	+3	+2
64	180	-1	+1
65	225	-1	0
66	270	-3	-3
67	315	+2	-1
68	360	+1	0

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3602								APE

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 APE The input type of stored pitch error compensation data is:
0: An incremental value.
1: A total value.

This function is effective to the following functions.
- Stored Pitch Error Compensation
- Bi-directional Pitch Error Compensation
- Interpolation Type Pitch Error Compensation
- Interpolation Type Straightness Compensation
- Spindle Command Synchronous Control Independent Pitch Error Compensation

> **NOTE**
> If this parameter is changed, the data of stored pitch error compensation is cleared automatically at next power on.

Warning

Massage	Description
DATA IS OUT OF RANGE	The pitch error compensation total value exceeds the range of –32768 to +32767. Input was denied.
DIFFERENCE WITH NEAR IS RANGE OVER	The difference between two consecutive data is exceeds the range of –128 to +127. Please modify the compensation data.

Alarm and message

Number	Massage	Description
PS0527	ILLEGAL DATA IN PITCH ERROR	In the effective pitch error compensation points that is decided by considering the setting of the parameters, the difference between two consecutive data exceeds the range of -128 to +127. Please correct the pitch error compensation data or change the parameter.
PS1300	ILLEGAL ADDRESS	The axis No. address was specified even though the parameter is not an axis–type while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. Axis No. cannot be specified in pitch error compensation data.

1.4 SETTINGS RELATED TO SERVO-CONTROLLED AXES

The servo interface features the following:

- Digitally controlled AC servo motor
- Motor feedback with serial Pulsecoders
  - (1) Absolute Pulsecoder with a resolution of 1,000,000 pulses/rev
  - (2) Absolute Pulsecoder with a resolution of 65,536 pulses/rev
  - (3) Incremental Pulsecoder with a resolution of 10,000 pulses/rev
- Scale feedback with A/B/Z signal interface

1.4.1 Parameters Related to Servo

Overview

Terms frequently used in explanation of parameters related to servo systems are listed below:

Least command increment.........The minimum unit of a command to be given from CNC to the machine tool

Detection unit............................The minimum unit which can detect the machine tool position

Command multiplier (CMR).....A constant to enable the weight of CNC command pulses to meet the weight of pulses from the detector

Detection multiplier (DMR)......A constant to enable the weight of CNC command pulses to meet the weight of pulses from the detector

> ⚠ **CAUTION**
> The relations among the least command increment, detection unit, CMR, and DMR are as specified below.
> Least command increment = CMR × detection unit
> Detection unit =
> Move amount per revolution of motor /
> (DMR × number of pulses of detector per revolution)
> The flexible feed gear function in the digital servo defines constant DMR using two parameters (Nos. 2084 and 2085) n and m (DMR = n/m).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1800							CVR	

[Input type] Parameter input
[Data type] Bit path

#1 CVR When velocity control ready signal VRDY is set ON before position control ready signal PRDY comes ON
0: A servo alarm is generated.
1: A servo alarm is not generated.

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx	APZx			OPTx	

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#1 OPTx The separate position detector is:
0: Not to be used (semi-closed system)
1: To be used (full-closed system)

> **NOTE**
> 1 In case of using the absolute position detector (bit 5 (APCx) of parameter No.1815 is set to 1), please set the following parameters that correspond to the absolute position detector which is actually used.
> - Parameter No.1815#6, No.1815#0, No.1817#3, No.1868, No.2275#1, No.2394
>
> If these parameters are not set correctly, the machine coordinates are not correctly established at power-on.
> 2 Set this parameter to 1 when using a linear scale with reference marks or a linear scale with distance-coded reference marks (serial) (full-closed system).

#4 APZx Machine position and position on absolute position detector when the absolute position detector is used
0: Not corresponding
1: Corresponding

When an absolute position detector is used, after primary adjustment is performed or after the absolute position detector is replaced, this parameter must be set to 0, power must be turned off and on, then manual reference position return must be performed. This completes the positional correspondence between the machine position and the position on the absolute position detector, and sets this parameter to 1 automatically.

#5 APCx Position detector
0: Other than absolute position detector
1: Absolute position detector (absolute Pulsecoder)

> **NOTE**
> 1 In case of using FANUC absolute pulsecoder as absolute position detector for the rotary axis A type that machine coordinate values are rounded in 0 to 360°, set bit 6 (RON) of parameter No. 1815 to 0. If the parameter is not set correctly, the machine coordinates are not correctly established at power-on.
> 2 In case of using a rotary scale without rotary data made by another company as absolute position detector, please refer to "CORRESPONDENCE OF ROTARY SCALE WITHOUT ROTARY DATA" in this manual and set an appropriate value corresponding to the detector. If the parameter is not set correctly, the machine coordinates are not correctly established at power-on.
> 3 When the servo axis is disabled (bit 4 (KSVx) of parameter No. 11802 is set to 1), an absolute position detector cannot be used (bit 5 (APCx) of parameter No. 1815 cannot be set to 1). If an absolute position detector is used, alarm SV0301, "APC ALARM; COMMUNICATION ERROR" is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
1816		DM3x	DM2x	DM1x				

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#4 DM1
#5 DM2
#6 DM3

By using DM1, DM2, and DM3, a detection multiplication factor (DMR) is set.
This parameter is valid when a separate position detector (AB phase) is used and parameters Nos. 2084 and 2085 are not set.

DM3	DM2	DM1	DMR
0	0	0	1/2
0	0	1	1
0	1	0	3/2
0	1	1	2
1	0	0	5/2
1	0	1	3
1	1	0	7/2
1	1	1	4

1820	Command multiplier for each axis (CMR)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] See below :

Set a command multiplier indicating the ratio of the least command increment to the detection unit for each axis.
Least command increment = detection unit × command multiplier

About relationship between the increment system and the least command increment, see Section "Increment System" in "SETTING EACH AXIS".

Setting command multiply (CMR), detection multiply (DMR), and the capacity of the reference counter

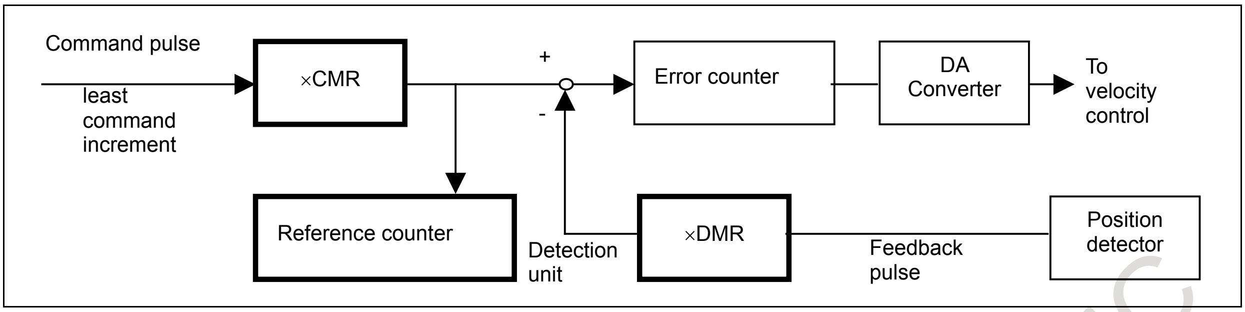


Fig. 1.4.1 (a)

Set CMR and DMR so that the pulse weight of + input (command from the CNC) into the error counter matches the pulse weight of -input (feedback from the position detector).

[Least command increment]/CMR=[Detection unit]=
[Feedback pulse unit]/DMR

[Least command increment]
Minimum unit of commands issued from the CNC to the machine

[Detection unit]
Minimum unit for machine position detection

The unit of feedback pulses varies, depending on the type of detector.

[Feedback pulse unit]=
[Amount of travel per rotation of the Pulsecoder]/
[Number of pulses per rotation of the Pulsecoder]

As the size of the reference counter, specify the grid interval for the reference position return in the grid method.

[Size of the reference counter]=[Grid interval]/[Detection unit]
[Grid interval]=[Amount of travel per rotation of the Pulsecoder]

The setting of a command multiplier is as follows:

(1) When command multiplier is 1 to 1/27
Set value = 1 / command multiplier + 100
Valid data range : 101 to 127

(2) When command multiply is 0.5 to 48
Set value = 2 × command multiplier
Valid data range : 1 to 96

> **NOTE**
> If a feedrate exceeding the feedrate found by the expression below is used, an incorrect travel amount may result or a servo alarm may be issued. Be sure to use a feedrate not exceeding the feedrate found by the following expression:
> Fmax[mm/min] = 196602 $\times 10^4 \times$ least command increment / CMR

1821	Reference counter size for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit

[Valid data range] 0 to 999999999

Set a reference counter size.

As a reference counter size, specify a grid interval for reference position return based on the grid method.

When a value less than 0 is set, the specification of 10000 is assumed.

1825	Servo loop gain for each axis

[Input type] Parameter input

[Data type] Word axis

[Unit of data] 0.01/sec

[Valid data range] 1 to 32767

Set the loop gain for position control for each axis.

When the machine performs linear and circular interpolation (cutting), the same value must be set for all axes. When the machine requires positioning only, the values set for the axes may differ from one another. As the loop gain increases, the response by position control is improved. A too large loop gain, however, makes the servo system unstable.

The relationship between the positioning deviation (the number of pulses counted by the error counter) and the feedrate is expressed as follows:

Positioning deviation = Feedrate / (60 × Loop gain)

Unit : Positioning deviation mm, inch or deg

Feedrate mm/min, inch/min, or deg/min

Loop gain 1/sec

1828	Positioning deviation limit for each axis in movement

[Input type] Parameter input

[Data type] 2-word axis

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.

If the positioning deviation exceeds the positioning deviation limit during movement, a servo alarm SV0411, “EXCESS ERROR (MOVING)” is generated, and operation is stopped immediately (as in emergency stop).

Generally, set the positioning deviation for rapid traverse plus some margin in this parameter. Refer to the following set value.

$$\textbf{Setting value} = \frac{\textbf{Rapid traverse rate}^{(\textbf{Note 1})}}{\textbf{60} \times \textbf{Servo loop gain}^{(\textbf{Note 2})}} \times \frac{1}{\textbf{Detection unit}} \times 1.2 \textbf{ to } 1.5$$

> ⚠ **CAUTION**
> If this parameter is not set correctly, machine or workpiece can be damaged.

> **NOTE**
> 1 Usually, this value is parameter No.1420. When the maximal feedrate of each axis exceeds parameter No.1420 according to the command and override, the value is the maximal feedrate of the each axis.
> 2 Usually, this value is parameter No.1825. When the servo loop gains other than parameter No.1825 is effective, the value is actual servo loop gains.

1829	Positioning deviation limit for each axis in the stopped state

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in the stopped state for each axis.
If, in the stopped state, the positioning deviation exceeds the positioning deviation limit set for stopped state, a servo alarm SV0410 is generated, and operation is stopped immediately (as in emergency stop).

1850	Grid shift and reference position shift for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -99999999 to 99999999

To shift the reference position, the grid can be shifted by the amount set in this parameter.
Up to the maximum value counted by the reference counter can be specified as the grid shift.
In case of bit 4 (SFDx) of parameter No. 1008 is 0: Grid shift
In case of bit 4 (SFDx) of parameter No. 1008 is 1: Reference point shift

> **NOTE**
> For setting the reference position without dogs, only the grid shift function can be used.
> (The reference position shift function cannot be used.)

1.4.2 Absolute Position Detection

Overview

An absolute position detector (absolute Pulsecoder) is an incremental Pulsecoder with an absolute counter. It detects the absolute position based on the value of the absolute counter. For an axis on which an absolute position detector is mounted, no reference position return is required at power-on because the machine position is always stored with batteries if the power to the CNC is turned off.
When the machine position has been brought into correspondence with the absolute position detector, the current position is read from the absolute counter at CNC power on and the machine and workpiece coordinate systems are automatically set using the value. In this case, you can immediately start automatic operation.
Restrictions described in the OPERATOR’S MANUAL and others that include those listed below are removed:

- "Reference position return is required after power-on."
- "The CNC can be used after reference position return is performed after power-on."

Explanation

- Parameter setting for the absolute position detector

Please set the following parameters that correspond to the absolute position detector.
Parameter No.1815#1, No.1815#6, No.1815#0, No.1817#3, No.1868, No.2275#1, No.2394

Please refer to "CORRESPONDENCE OF ROTARY SCALE WITHOUT ROTARY DATA" in this manual.

> ⚠ **WARNING**
> Set the parameters that correspond to the absolute position detector which is actually used. If the parameter is not set correctly, the machine coordinates are not correctly established at power-on.

1. Linear axis

	Parameter No.						
	1815#1	**1815#6**	**1815#0**	**1817#3**	**1868**	**2275#1**	**2394**
Semi-closed feedback loop	0	0	0	0	0	0	0
Full-closed feedback loop	1	0	0	0	0	0	0

2. Rotary axis (A type)

	Parameter No.						
	1815#1	**1815#6**	**1815#0**	**1817#3**	**1868**	**2275#1**	**2394**
Semi-closed feedback loop (a scale with rotary data)	0	0	0	0	0	0	0
Semi-closed feedback loop (a scale without rotary data)	0	1	0	0	0	*1	*1
Full-closed feedback loop (a scale with rotary data)	1	0	0	0	0	0	0
Full-closed feedback loop (a scale without rotary data)	1	1	0	0	0	*1	*1

3. Rotary axis (B type)

	Parameter No.						
	1815#1	**1815#6**	**1815#0**	**1817#3**	**1868**	**2275#1**	**2394**
Semi-closed feedback loop (a scale with rotary data)	0	0	0	0	0	0	0
Semi-closed feedback loop (a scale without rotary data)	0	0	*2	*2	*2	*1	*1
Full-closed feedback loop (a scale with rotary data)	1	0	0	0	0	0	0
Full-closed feedback loop (a scale without rotary data)	1	0	*2	*2	*2	*1	*1

*1 These parameters are set according to the detector which is actually used. For details, please refer to "CORRESPONDENCE OF ROTARY SCALE WITHOUT ROTARY DATA".

*2 These parameters are set according to movable range of rotary axis. For details, please refer to "CORRESPONDENCE OF ROTARY SCALE WITHOUT ROTARY DATA".

- **About a rotary data in detector.**

The rotary data is number of revolution which is counted up or down per one rotation.
There are both "a scale with rotary data" and "a scale without rotary data" by detector.

- A scale with rotary data

  For detector manufactured by FANUC, all detectors are “a scale with rotary data”.
  Example of detector which manufactured by FANUC

  -Semi-closed system
  - When a detector with αi servo motor (αi Pulsecoder) is used.
  - When αiCZ sensors is used with a synchronous built-in servo motor.

  - Full-closed system
  - When αiCZ sensors is used as a separate detector.

- When αA1000S sensors which is separate pulsecoder is used as a separate detector.

- A scale without rotary data

For the following detectors, these are "a scale without rotary data".
The following detectors are not all of the detector which is "a scale without rotary data".

- HEIDENHAIN
  RCN223F / RCN727F
- Magnescale
  RU77
- MITSUBISHI HEAVY INDUSTRIES
  MPRZ Series
- RENISHAW
  RESOLUTE (rotary encoder)

NOTE
Please make sure to confirm the using detector to the detector manufacturers.

- Coordinate systems at power-on

(1) The machine and workpiece coordinate systems are automatically set. When bit 3 (PPD) of parameter No. 3104 is set to 1, the relative position display is preset. When parameter PPD is set to 0, it is also possible to preset relative coordinates with machine coordinates by setting bit 5 (PWR) of parameter No. 11277 to 1.
(2) The amount of shift by coordinate system setting (G92 for the M series or G50 for the T series) or local coordinate system setting (G52) is cleared.

- Setting the zero point of an absolute position detector

In the following cases, set the zero point of an absolute position detector (bring the counter value of the absolute position detector into correspondence with the reference position):

- Primary adjustment is performed (after the reference position is determined).
- The reference position is changed.
- An absolute position detector is replaced with another.
- A servo motor is replaced with another.
- Alarm DS0300 is issued.
- The file memory is replaced with another.
- Parameter data is cleared.

To set the zero point of an absolute position detector, the following two methods are available: One method using manual reference position return and the other using MDI operation.

- Manual reference position return

Follow the procedure below to perform manual reference position return.
(1) Set bit 4 (APZ) of parameter No. 1815 to 0. Alarms PW0000 and DS0300 are displayed.
(2) Turn the power off, then on again. Alarm DS0300 is displayed.
(3) Perform manual reference position return. When manual reference position return is completed, bit 4 (APZ) of parameter No. 1815 is automatically set to 1.
(4) Press the reset button to release alarm DS0300.
Operation in step (3) can also be performed by reference position setting without DOG, reference point setting with mechanical stopper, or reference point setting with mechanical stopper by grid method.

- Setting the zero point using MDI operation

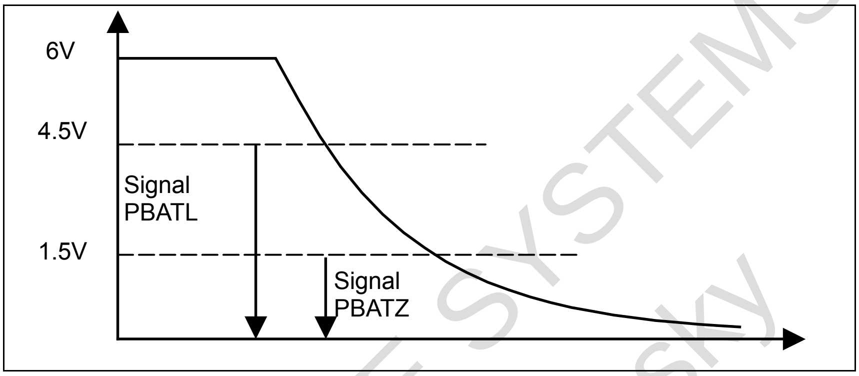
Follow the procedure below to set the zero point using MDI operation.
(1) Set bit 4 (APZ) of parameter No. 1815 to 0. Alarms PW0000 and DS0300 are displayed.
(2) Turn the power off, then on again. Alarm DS0300 is displayed.
(3) Move the tool to the reference position in jog, manual handle, or manual incremental feed.
(4) Set bit 4 (APZ) of parameter No. 1815 to 1. Alarm PW0000 is displayed.

(5) Turn the power off, then on again.

> ⚠ **CAUTION**
> If setting the zero point using MDI operation causes the reference position to be lost for some reason, any FANUC service personnel and end user cannot restore the reference position accurately. In this case, you should ask the machine tool builder to restore the accurate reference position, which requires an immense amount of time for restoration. So, do not use MDI operation to set the zero point. For details, see "Reexecuting zero point setting" described later.

- Voltage of backup batteries

Normally, the voltage of the backup batteries for the absolute position detector is 6 V. The voltage drops as time goes on.



When the voltage of the backup batteries becomes 4.5 V or less

- When the voltage of the backup batteries becomes 4.5 V or less, "APC" is highlighted and blinks in the state display area on the screen, and absolute position detector battery voltage low alarm signal PBATL <Fn172.7> is set to "1".
- When the power to the CNC is turned off, then on again without replacing the backup batteries and the counter value data of the absolute position detector is retained, alarm DS0307, "APC ALARM: BATTERY LOW 1" is displayed. Signal PBATL remains set to "1".
- In this status, when emergency stop is canceled or the reset key is pressed, alarm DS0307 is released, but "APC" is highlighted and blinks in the state display area on the screen. Signal PBATL remains set to "1".
- When the batteries are replaced with ones the voltage of which is higher than or equal to the rating and the reset key is pressed, "APC" in the status display area on the screen goes out and signal PBATL is set to "0".

> ⚠ **CAUTION**
> If a battery voltage low alarm is displayed, replace the batteries with new ones immediately. If the batteries are not replaced, the counter value data of the absolute position detector is lost.

When the voltage of the backup batteries becomes 1.5 V or less

- When the voltage of the backup batteries is 1.5 V or less, if the power to the CNC is turned off, the battery voltage is assumed to be 0, and the counter value data of the absolute position detector is lost. When the power to the CNC is turned on again, alarm DS0300, "APC ALARM: NEED REF RETURN" and alarm DS0306, "APC ALARM: BATTERY VOLTAGE 0" are displayed, and absolute position detector battery voltage zero alarm signal PBATZ <Fn172.6> is set to "1".
  At this time, bit 4 (APZ) of parameter No. 1815, which indicates that the correspondence between the machine position and the position on the absolute position detector is established, is set to 0.

- In this status, when emergency stop is canceled or the reset key is pressed, only alarm DS0306 is released. Signal PBATZ remains set to “1”.
- Follow the procedure below to set the zero point of the absolute position detector.
  1 Replace the batteries with ones the voltage of which is higher than or equal to the rating.
  2 Rotate the motor manually at least one turn.
  3 Turn the power to the CNC and servo amplifier off.
  4 Turn the power to the CNC and servo amplifier on.
  5 Set the zero point of the absolute position detector. After that, press the reset key to release alarm DS0300.
- When the power to the CNC is turned off, then on again after the batteries are replaced with ones the voltage of which is higher than or equal to the rating, signal PBATZ is set to “0”.

- Reexecuting zero point setting

If any of the following states occurs after the zero point of an absolute position detector is set, the reference position is lost, which requires the reexecution of zero point setting:

- An event occurred, which set any bit of diagnosis data Nos. 310 and 311 to 1. (For example, the voltage of the backup batteries became 1.5 V or less, or a parameter was updated, which caused the reference position to be lost.)
- A failure occurred in the pulse coder or motor, and it was replaced.

In this case, if the zero point is set using MDI operation, any FANUC service personnel and end user cannot restore the reference position accurately, and you should ask the machine tool builder to restore the accurate reference position, which requires an immense amount of time for restoration. So, do not use MDI operation to set the zero point. The following table lists other methods for setting the zero point to restore the reference position.

Table 1.4.2 (a) Reference position restoration

Zero point setting method	Overview	Advantages	Disadvantages
Manual reference position return	Uses the grid position created by the detector and a deceleration dog to establish the reference position.	Enables the reference position to be restored most reliably.	Requires a deceleration dog.
Reference position setting without DOG	Uses the grid position created by the detector and starts reference position return operation near the reference position to establish the reference position without dogs.	- Requires no deceleration dog. - Allows the reference position to be restored easily when the position at which to start restoring the reference position is clear.	Cannot restore the reference position if the position at which to start restoring the reference position is not clear.
Reference point setting with mechanical stopper	Butts the tool against the mechanical stopper along an axis and makes the tool withdraw a set distance from the mechanical stopper to establish the reference position.	- Requires no deceleration dog. - Allows the reference position to be restored very easily.	Lacks the positional accuracy of the reference position when the following factors are taken into account: Deformation of the part where the tool is butted, foreign matters attached to the part, etc.

Zero point setting method	Overview	Advantages	Disadvantages
Reference point setting with mechanical stopper by grid method	Butts the tool against the mechanical stopper along an axis and makes the tool withdraw to the grid position created by the detector to establish the reference position.	- Requires no deceleration dog. - Allows the reference position to be restored very easily. - Enables the accurate reference position to be established since the grid position is used.	

> ⚠ **CAUTION**
> 1 Do not use MDI operation to set the zero point. It is impossible to restore the reference position accurately using this method.
> 2 Use manual reference position return using a grid whenever possible.
> 3 To use reference position setting without DOG, be sure to place the eye mark indicating the start position of reference position return at a position where it can be seen without removing the cover and other parts of the machine and make the mark last after long-time machine operation.
> 4 To use reference point setting with mechanical stopper with using no grid, design the machine so that the part where the tool is butted is not deformed and is free from chips and other foreign matters.
> 5 Be sure to attach a manual which details the method for reference position return along each axis to the machine.

- Rotary axis A type

When the increment system (bit 0 to 1 (ISA or ISC) of parameter No. 1013) is changed for an axis in the following status, bit 4 (APZ) of parameter No. 1815 is set to 0 to set the status of the correspondence between the machine position and the position on the absolute position detector to unestablished:

- The zero point is established using the absolute position detector. (Bits 4 (APC) and 5 (APZ) of parameter No. 1815 = 1)
- Rotary axis A type (Bit 0 (ROT) of parameter No. 1006 = 1, Bit 1 (ROS) of parameter No. 1006 = 0)
- Bit 6 (RON) of parameter No. 1815 = 0

By setting the bit to 0, alarm DS0300 is issued and the zero point of the absolute position detector must always be set again, which prevents malfunction caused if zero point setting is not performed again.
At this time, bit 0 (PR1) of diagnosis data No. 310 is set to 1.

- Information of setting the zero point

The method of setting the zero point can be confirmed by diagnosis data No.3520.

- When the servo axis is disabled (bit 4 (KSVx) of parameter No. 11802 is set to 1)

An absolute position detector cannot be used (bit 5 (APCx) of parameter No. 1815 cannot be set to 1).
If an absolute position detector is used, alarm SV0301, "APC ALARM; COMMUNICATION ERROR" is issued.

Signal

Absolute position detector battery voltage low alarm signal PBATL<Fn172.7>

[Classification] Output signal
[Function] Notifies that the life of the absolute position detector battery is about to expire.
[Operation] These signal is “1” in the following case:

- The battery voltage for the absolute position detector becomes lower than or equal to the rating. The battery need be replaced in the immediate future.

These signal is “0” in the following case:

- The battery voltage for the absolute position detector is higher than or equal to the rating.

Absolute position detector battery voltage zero alarm signal PBATZ<Fn172.6>

[Classification] Output signal

[Function] Notifies that the life of the absolute position detector battery has expired.

[Operation] These signals are “1” in the following case:

- The batteries for the absolute position detector have run out. The batteries need be replaced in the status in which the power to the machine is on.

These signals are “0” in the following case:

- The batteries for the absolute position detector feed the voltage required for data backup.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn172	PBATL	PBATZ						

Diagnosis data

Why bit 4 (APZ) of parameter No. 1815 is set to 0 can be checked using diagnosis data Nos. 310 and 311. After any bit of diagnosis data Nos. 310 and 311 is set to 1, it remains set to 1 until the zero point of the absolute position detector on the axis is set. Each bit of diagnosis data Nos. 310 and 311 is described below with its corresponding cause.

	#7	#6	#5	#4	#3	#2	#1	#0
310		DTH	ALP		BZ2	BZ1	PR2	PR1

PR1 One of the following parameters were changed:
No.1815#0, No.1815#1, No.1815#6, No.1817#3, No.1820, No.1821, No.1822, No.1823, No.1850, No.1868, No.1869, No.1874, No.1875, No.1876, No.1878, No.1883, No.1884, No.2022, No.2084, No.2085, No.2179, increment system for a rotary axis A type (see “Explanation” for details.), No.11802#6, No.11810

PR2 The value of bit 1 (ATS) of parameter No. 8303 was changed. Alternatively, when bit 7 (SMA) of parameter No. 8302 was set to 1, APZ of the relevant axis for synchronous operation became 1.

BZ1 A battery voltage of 0 V was detected. (Inductosyn)

BZ2 A battery voltage of 0 V was detected. (separate position detector)

ALP Although the α Pulsecoder did not indicate one turn, the zero point was set using MDI operation.
Alternatively, the CNC could not obtain a valid value from the absolute Pulsecoder.

DTH An axis was released from control by using controlled axis detach signal DTCH <Gn124> or bit 7 (RMV) of parameter No. 0012.

	#7	#6	#5	#4	#3	#2	#1	#0
311		DUA	XBZ	GSG	AL4	AL3	AL2	AL1

AL1 SV alarms SV0301 to SV0305 were issued.

AL2 When bit 1 (CRF) of parameter No. 1819 was 1, soft disconnect alarm SV0445 or SV0447 or abnormal analog signal alarm SV0646 was detected.

AL3 A battery voltage of 0 V was detected. (built-in serial Pulsecoder)

AL4 Count miss alarm SV0367 was detected.

GSG Disconnection alarm ignore signal NDCAL <Gn202> was changed from “1” to “0”.

XBZ A battery voltage of 0 V or count miss alarm SV0382 was detected. (separate position detector installed for each serial method)

DUA When bit 1 (CRF) of parameter No. 1819 was 1, excess error (semi-full) alarm SV0421 was detected.

If a battery low alarm is issued, the cause of the issuance can be checked with diagnosis data No. 3019.

	#7	#6	#5	#4	#3	#2	#1	#0
3019			EXP	INP	ABP			

[Data type] Axis

ABP Battery low in the phase A/B

ANP Battery low in the serial Pulsecoder (built-in position detector)

EXP Battery low in the serial separate position detector

3520	Information of setting the zero point for absolute position detection

[Data type] Byte axis

[Unit of data] None

[Valid data range] 0 to 3

To set the zero point of absolute position detection:

0 : is not performed yet.

1 : was performed by the manual reference position return.

2 : was performed by MDI operation.

3 : was performed by the reading of parameter file.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1803	NFP							

[Input type] Parameter input

[Data type] Bit path

#7 NFP If position matching between the machine position and absolute position detector is not performed even once, follow-up operation is:

0: Not performed.

1: Performed.

	#7	#6	#5	#4	#3	#2	#1	#0
1815		RONx	APCx	APZx			OPTx	

[Input type] Parameter input

[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#1 OPTx The separate position detector is:

0: Not to be used (semi-closed system)

1: To be used (full-closed system)

> **NOTE**
> In case of using the absolute position detector (bit 5 (APCx) of parameter No.1815 is set to 1), please set the following parameters that correspond to the absolute position detector which is actually used.
> - parameter No.1815#6, No.1815#0, No.1817#3, No.1868, No.2275#1, No.2394
>
> If these parameters are not set correctly, the machine coordinates are not correctly established at power-on.

#4 APZx Machine position and position on absolute position detector when the absolute position detector is used

0: Not corresponding
1: Corresponding

When an absolute position detector is used, after primary adjustment is performed or after the absolute position detector is replaced, this parameter must be set to 0, power must be turned off and on, then manual reference position return must be performed. This completes the positional correspondence between the machine position and the position on the absolute position detector, and sets this parameter to 1 automatically.

#5 APCx Position detector

0: Other than absolute position detector
1: Absolute position detector (absolute Pulsecoder)

> **NOTE**
> 1 In case of using FANUC absolute pulsecoder as absolute position detector for the rotary axis A type that machine coordinate values are rounded in 0 to 360°, set bit 6 (RON) of parameter No.1815 to 0. If the parameter is not set correctly, the machine coordinates are not correctly established at power-on.
> 2 In case of using a rotary scale without rotary data made by another company as absolute position detector, please refer to "CORRESPONDENCE OF ROTARY SCALE WITHOUT ROTARY DATA" in this manual and set an appropriate value corresponding to the detector. If the parameter is not set correctly, the machine coordinates are not correctly established at power-on.
> 3 When the servo axis is disabled (bit 4 (KSVx) of parameter No. 11802 is set to 1), an absolute position detector cannot be used (bit 5 (APCx) of parameter No. 1815 cannot be set to 1). If an absolute position detector is used, alarm SV0301, "APC ALARM; COMMUNICATION ERROR" is issued.

#6 RONx With a rotary axis A type, a rotary encoder for detecting an absolute position within one revolution is:

0: Not used.
1: Used.

NOTE
1. This parameter is available for only the rotary axis A type with an absolute position detector (absolute Pulsecoder).This function cannot be used for a rotary scale with distance-coded reference marks (serial) or for a distance coded rotary scale interface (phase A/B).
2. Set this parameter to 1 for the rotary axis A type using a rotary scale without rotary data such as HEIDENHAIN rotary scale RCN 223F, 727F, etc.
   Set this parameter to 0 in any other case. If the parameter is not set correctly, the machine coordinates are not correctly established at power-on.
3. When this parameter is set, machine position and position on absolute position detector become uncorresponding. Consequently, the bit 4 (APZ) parameter No. 1815 (indicating that the correspondence is established) is set to 0, and alarm DS0300 is issued. Why the parameter bit 4 (APZ) parameter No. 1815 is set to 0 can be checked using bit 0 of diagnosis data No. 310.

	#7	#6	#5	#4	#3	#2	#1	#0
1817					SCRx			

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#3 SCRx Specifies whether to convert scale data by using threshold position (parameter No. 1868) so that rotary axis B type is available, in the case of the axis B type that use a rotary scale without data (the number of rotation), whose movable range is under one rotation:
0 : Not to convert.
1 : To convert.

NOTE
1. This parameter is available for only the rotary axis B type with an absolute position detector (absolute Pulsecoder) or a rotary scale with distance-coded reference marks (serial). This function cannot be used for distance coded rotary scale interface (phase A/B).
2. Don't set this parameter in the case of no uncontinuous point within movable range of rotary axis even if the rotary axis B type.
3. When this parameter is set, machine position and position on absolute position detector become uncorresponding. Consequently, the bit 4 (APZ) of parameter No. 1815 (indicating that the correspondence is established) is set to 0, and alarm DS0300 is issued. Why the bit 4 (APZ) of parameter No. 1815 is set to 0 can be checked using bit 0 of diagnosis data No. 310.

NOTE

4. This function cannot be used together with the bit 0 (RVSx) of parameter No. 1815 = 1 that save rotary data by CNC, in the case of a rotary axis B type whose movable range is over one rotation.
5. In this function, the amount of one rotation of rotary axis assumes 360, and the machine position 0 assumes the reference position. It is not possible to apply to a rotary axis other than the above-mentioned setting.
6. Set the parameter No. 1240 to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
1819							CRFx	

[Input type] Parameter input
[Data type] Bit axis

#1 CRFx When the servo alarm SV0445 (soft disconnection), SV0447 (hard disconnection (separate)), or SV0421 (dual position feedback excessive error) is issued:
0: The reference position established state is not affected.
1: The reference position unestablished state is assumed. (Bit 4 (APZ) of parameter No. 1815 is set to 0.)

1860	Value 1 for the zero point of the absolute position detector
1861	Value 2 for the zero point of the absolute position detector
1862	Value 3 for the zero point of the absolute position detector

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Locked parameter
[Data type] 2-word axis

Parameters Nos.1860 to 1862 are values for the zero point of the absolute position detector (association with reference position and the counter value of the absolute position detector).
The CNC automatically sets parameters Nos.1860 to 1862 when correspondence between the reference position and the absolute position detector is performed.
It is not available to set parameters Nos.1860 to 1862 by using inputting of MDI operation, FOCAS2, PMC window or programmable parameter input(G10).

The zero point data of the absolute position detection can be restored and parameters Nos.1860 to 1862 can be set, by the parameter input from connected I/O unit or by restoring the SRAM area of boot system.
When the zero point data of the absolute position detection are restored, be careful about the following.

- When the zero point data of absolute position detection are restored, back up them right before the restoring and use backup data.
- When the motor or the detector is exchanged, the zero point data of absolute position detection cannot be restored. After exchange, set the zero point of the absolute position detector again.

- When the zero point data of the absolute position detection are restored by them of the other machine, the zero point of the absolute position detector is set to wrong position. Don't restore the zero point data of absolute position detection by them of the other machine.
- When the zero point data of absolute position detection are backed up with reference position not established and restored by backup data, setting of the zero point of the absolute position detector is required after restoring.

> ⚠ **WARNING**
> If the zero point of the absolute position detector is not set properly, the machine coordinate system is not established correctly.

1868	Threshold position for converting scale data (each axis)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (Refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

In the case that scale data of a rotary scale without rotary data is larger than the scale data of the threshold position (this parameter value), it is converted to be continuous data in movable range by subtracting data of one rotation. The position out of movable range (angle from an uncontinuous point) must be set as threshold position. As for the axis for which this parameter is set to 0, conversion of scale data is not performed.

> **NOTE**
> 1 This parameter is available for only the rotary axis B type with an absolute position detector (absolute Pulsecoder) or a rotary scale with distance-coded reference marks (serial), for which the bit 3 (SCRx) of parameter No. 1817 is set to 1.This function cannot be used for distance coded rotary scale interface (phase A/B).
> 2 This function cannot be used for distance coded rotary scale interface (phase A/B).
> 3 Don't set this parameter in the case of no uncontinuous point within movable range of rotary axis even if the rotary axis B type.
> 4 When this parameter is set, machine position and position on absolute position detector become uncorresponding. Consequently, the bit 4 (APZ) of parameter No. 1815 (indicating that the correspondence is established) is set to 0, and alarm DS0300, "APC ALARM: NEED REF RETURN" is issued. Why the bit 4 (APZ) of parameter No. 1815 is set to 0 can be checked using bit 0 of diagnosis data No. 310.

	#7	#6	#5	#4	#3	#2	#1	#0
2275							RCNCLR	

[Input type] Parameter input
[Data type] Bit axis

#1 RCNCLR The number of revolution is:
0: Not to be cleared.
1: To be cleared. (To use the RCN220, RCN223, or RCN723, set the bit to 1.)
This function bit is to be set in combination with the number of data mask digits, described in parameter No.2394.

> **NOTE**
> This parameter is set according to the detector which is actually used. For details, please refer to "CORRESPONDENCE OF ROTARY SCALE WITHOUT ROTARY DATA".

2394	Number of data mask digits

[Input type] Parameter input
[Data type] Word axis
[Setting value] 5 or 8
The value to be set in this parameter depends on the detector. At present, only the following detectors require clearing the speed data. Set 5 to use the RCN220, and set 8 to use the RCN223, or RCN723. This parameter is to be set in combination with bit 1 (RCNCLR) of parameter No.2275.

> **NOTE**
> 1 The rotary data of the RCN220, RCN223, or RCN723 is maintained while the power to the separate detector interface unit is on. The data, however, is cleared when the unit is turned off.
> Since the rotary data becomes undetermined depending on where the power is turned off, it is necessary to make a setting to clear the rotary data. In addition, for this reason, the RCN220,RCN223, and RCN723 cannot be used with a linear axis.
> However, the detectors are available with a linear axis even if the rotary data is cleared by setting bit 6 (RVL) of parameter No.11802 is set to 1 and bit 0 (RVS) of parameter No.1815 is set to 1.
> 2 This parameter is set according to the detector which is actually used. For details, please refer to "CORRESPONDENCE OF ROTARY SCALE WITHOUT ROTARY DATA".

	#7	#6	#5	#4	#3	#2	#1	#0
11277			PWR					

[Input type] Parameter input
[Data type] Bit path

#5 PWR When a coordinate system is set at power-on using an absolute position detector (absolute pulse coder) with bit 3 (PPD) of parameter No. 3104 set to 0:
0: The axis is preset with 0.
1: The axis is preset with machine coordinates.

	#7	#6	#5	#4	#3	#2	#1	#0
11802				KSVx				

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#4 KSVx Servo axis is:
0: Enabled.
1: Disabled.

> **NOTE**
> 1 This setting is effective regardless of the value of parameter No. 1023.
> 2 If this setting is made for the axis subject to Cs axis contour/spindle positioning, Cs axis contour/spindle positioning will be disabled.
> 3 When the servo axis is disabled (bit 4 (KSVx) of parameter No. 11802 is set to 1), an absolute position detector cannot be used (bit 5 (APCx) of parameter No. 1815 cannot be set to 1). If an absolute position detector is used, alarm SV0301, "APC ALARM; COMMUNICATION ERROR" is issued.

Alarm and message

Number	Message	Description
PS0090	REFERENCE RETURN INCOMPLETE	1. The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return. 2. An attempt was made to set the zero position for the absolute position detector by return to the reference position when it was impossible to set the zero point. Rotate the motor manually at least one turn, and set the zero position of the absolute position detector after turning the CNC and servo amplifier off and then on again.
DS0300	APC ALARM: NEED REF RETURN	A setting to zero position for the absolute position detector (association with reference position and the counter value of the absolute position detector) is required. Perform the return to the reference position. This alarm may occur with other alarms simultaneously. In this case, other alarms must be handled first.
DS0306	APC ALARM: BATTERY VOLTAGE 0	The battery voltage of the absolute position detector has dropped to a level at which data can no longer be held. Or, the power was supplied to the Pulsecoder for the first time. The battery or cable is thought to be defective. Replace the battery with the machine turned on.
DS0307	APC ALARM: BATTERY LOW 1	The battery voltage of the absolute position detector has dropped to a level at which a replacement is required. Replace the battery with the machine turned on.
DS0308	APC ALARM: BATTERY LOW 2	The battery voltage of the absolute position detector dropped to a level at which a replacement was required in the past. (including during power off) Replace the battery with the machine turned on.

Number	Message	Description
DS0309	APC ALARM: REF RETURN IMPOSSIBLE	An attempt was made to set the zero point for the absolute position detector by MDI operation when it was impossible to set the zero point. Rotate the motor manually at least one turn, and set the zero position of the absolute position detector after turning the CNC and servo amplifier off and then on again.
SV0301	APC ALARM: COMMUNICATION ERROR	1. Since the absolute-position detector of the phase A/B caused a communication error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective. 2. When the servo axis is disabled, an absolute position detector cannot be used. Check the setting of bit 5 (APCx) of parameter No. 1815 and bit 4 (KSVx) of parameter No. 11802.
SV0302	APC ALARM: OVER TIME ERROR	Since the absolute-position detector of the phase A/B caused an overtime error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0303	APC ALARM: FRAMING ERROR	Since the absolute-position detector of the phase A/B caused a framing error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0304	APC ALARM: PARITY ERROR	Since the absolute-position detector of the phase A/B caused a parity error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0305	APC ALARM: PULSE ERROR	Since the absolute-position detector of the phase A/B caused a pulse error, the correct machine position could not be obtained. The absolute-position detector, or cable is thought to be defective.
SV0306	APC ALARM: OVER FLOW ERROR	Since the amount of positional deviation overflowed, the correct machine position could not be obtained. Check to see the parameters Nos. 2084 or 2085.
SV0307	APC ALARM: MOVEMENT EXCESS ERROR	Since the machine moved excessively, the correct machine position could not be obtained.

Caution

⚠ CAUTION

For an absolute position detector, batteries are used because the absolute position must be retained. When the battery voltage becomes low, a battery low alarm for the absolute position detector is displayed on the machine operator's panel or screen. If a battery voltage low alarm is displayed, replace the batteries with new ones immediately. If the batteries are not replaced, the absolute position data in the absolute position detector is lost.

Note

NOTE

1 For the procedure for replacing batteries, refer to the method for replacing batteries in "MAINTENANCE" in the OPERATOR'S MANUAL.
2 When replacing batteries with new ones after alarm DS0306, "APC ALARM: BATTERY VOLTAGE 0" is issued, manually rotate the motor at least one turn and turn the power to the CNC and servo amplifier off, then on again. Then, set the zero point of the absolute position detector (using manual reference position return or MDI operation).
If the above operation is not performed, alarm PS0090 or DS0309 is issued.
3 Do not set parameters Nos. 1860 to 1862 because the CNC automatically sets them when the zero point of an absolute position detector is set (the counter value of the absolute position detector is brought into correspondence with the reference position).

1.4.3 FSSB Setting

Overview

Connecting the CNC control section to servo amplifiers and spindle amplifiers via a high-speed serial bus (FANUC Serial Servo Bus, or FSSB), which uses only one fiber optics cable, can significantly reduce the amount of cabling in machine tool electrical sections.
In a system using the FSSB, it is necessary to set up the following parameters to specify its axes.

Table 1.4.3 (a) Parameters related to FSSB

Parameter No.	Contents
1023	Number of the servo axis for each axis
1902#1	Whether FSSB automatic setting is not completed (0) or completed (1)
2013#0	Whether HRV3 current loop is not used (0) or used (1)
3717	Spindle amplifier number of each spindle
11802#4	Whether the servo axes are enabled (0) or disabled (1)
24000 to 24095	ATR value corresponding to each slave on each FSSB line
24096 to 24103	Connector number of each separate detector interface unit

These parameters can be specified using the following three methods:

1. Manual setting 1
   Parameters are defaulted according to the setting of parameters Nos. 1023 and 3717. There is no need to specify parameters Nos. 24000 to 24095 and 24096 to 24103. No automatic setting is used. Note that some functions are unusable.

2. Automatic setting
   FSSB-related parameters are automatically specified by entering FSSB-related information on the FSSB setting screen.

3. Manual setting 2
   FSSB-related parameters are directly entered.

Explanation

- Slave

In an FSSB-based system, a fiber optics cable is used to connect the CNC to servo amplifiers or spindle amplifiers and separate detector interface units (hereafter referred to as separate detectors). These amplifiers and separate detectors are called slaves. The two-axis amplifier consists of two slaves, and the three-axis amplifier consists of three slaves. Slave numbers 1, 2 to 32 are sequentially assigned to slaves for each FSSB line. Number 1 is assigned to the slave nearest to the CNC.

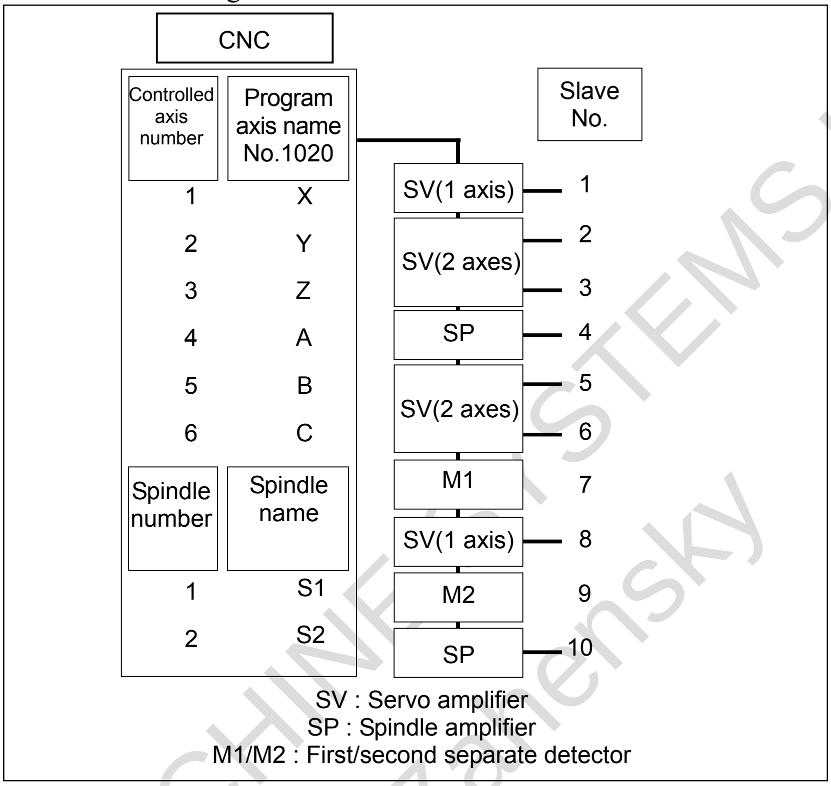


Fig. 1.4.3 (a) Slave

- Manual setting 1

The manual setting 1 is valid when the following parameter have the following values:

Bit 0 (FMD) of parameter No. 1902 = 0
Bit 1 (ASE) of parameter No. 1902 = 0

By manual setting 1, the values set for parameters Nos. 1023 and 3717 are used for setting slave numbers at power-on. Specifically, an axis for which parameter No. 1023 is set to 1 is connected to the amplifier nearest to the CNC, while an axis for which parameter No. 1023 is set to 2 is the second one from the CNC. A spindle for which parameter No. 3717 is set to 1 is connected to the spindle amplifier nearest to the CNC, while a spindle for which parameter No. 3717 is set to 2 is the Second one from the CNC.

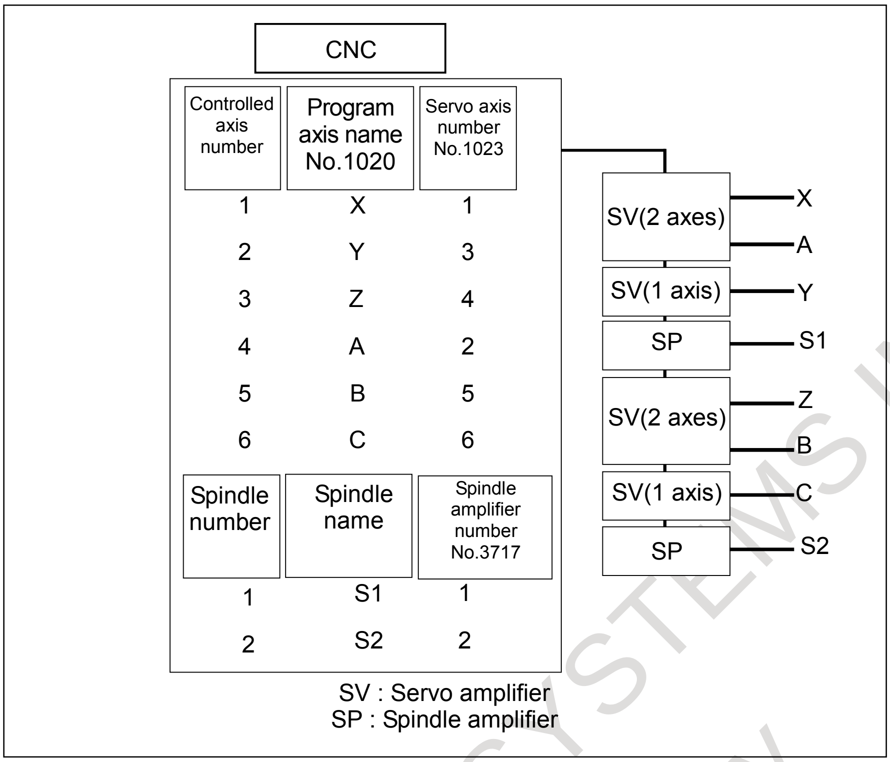


Fig. 1.4.3 (b) Setting example (manual setting 1)

By manual setting 1, some of the following functions and values cannot be used, as described below. To use the following functions and values, use automatic setting or manual setting 2.

- No separate detector can be used.
- No number can be skipped in parameter No. 1023 (other than the unavailable servo axis numbers, multiple of 8 and value obtained by subtracting 1 from a multiple of 8). For example, if servo axis number 3 is set for an axis without setting servo axis number 2 to any axis, manual setting 1 cannot be performed.
- No number can be skipped in parameter No. 3717 (spindle amplifier number of each spindle). For example, if spindle amplifier number 3 is set for a spindle without setting spindle amplifier number 2 to any spindle, manual setting 1 cannot be performed.
- The following servo functions cannot be used:
  - Servo HRV3 control
  - Tandem control
  - Electronic gear box (EGB)

- Automatic setting

Automatic setting can be used on the FSSB setting screen, if the following parameter is set as follows:
Bit 0 (FMD) of parameter No. 1902 = 0

On the FSSB setting screen, automatic setting should be enabled by means of the following procedure:

1 Make settings on the servo amplifier setting screen.
2 Make settings on the spindle amplifier setting screen.
3 Make settings on the axis setting screen.
4 Press the soft key [SETTING] to perform automatic setting. If setting data contains an error, a warning message is displayed. Set correct data again.

In this way, FSSB-related parameters are set. When each parameter has been set up, bit 1 (ASE) of parameter No. 1902 is set to 1. Switching the power off then back on again causes FSSB setting to be performed according to these parameter settings.
For details of the FSSB setting screen, see the FSSB data display and setting procedure, described below.

NOTE

1 Set the following items before executing FSSB automatic setting:
  - Set parameter No. 1020 (program axis name for each axis).
  - If you use the electronic gear box (EGB) function, set bit 0 (SYN) of parameter No. 2011 to 1 for the EGB slave axis and EGB dummy axis.

2 There are the following restrictions on FSSB automatic setting:
  - The current loop (HRV) is common to all axes.
  - Since any servo axis number is not skipped when parameter No. 1023 is set, the numbers cannot be set so that a specific servo axis number is skipped.

3 FSSB automatic setting cannot be performed when at least one of the following settings is made:
  - The FSSB setting mode is the manual setting 2 mode (bit 0 (FMD) of parameter No. 1902 is set to 1).
  - The serial feedback dummy function is enabled (bit 0 (DMY) of parameter No. 2009 is set to 1).
  - A servo axis is disabled for all axes (bit 4 (KSV) of parameter No. 11802 is set to 1).
  - One connector on an separate detector interface unit is used for more than one axes (bit 5 (SSC) of parameter No. 14476 is set to 1).

- Manual setting 2

After bit 0 (FMD) of parameter No. 1902 is set to 1 or automatic setting has been terminated (bit 1 (ASE) of parameter No. 1902 is set to 1), manual setting 2 for each parameter for axis setting can be performed. To perform manual setting 2, set parameters related to FSSB. Refer to the Parameter Manual for the definition of each parameter.

Example of setting parameters

- When separate detector interface units are connected

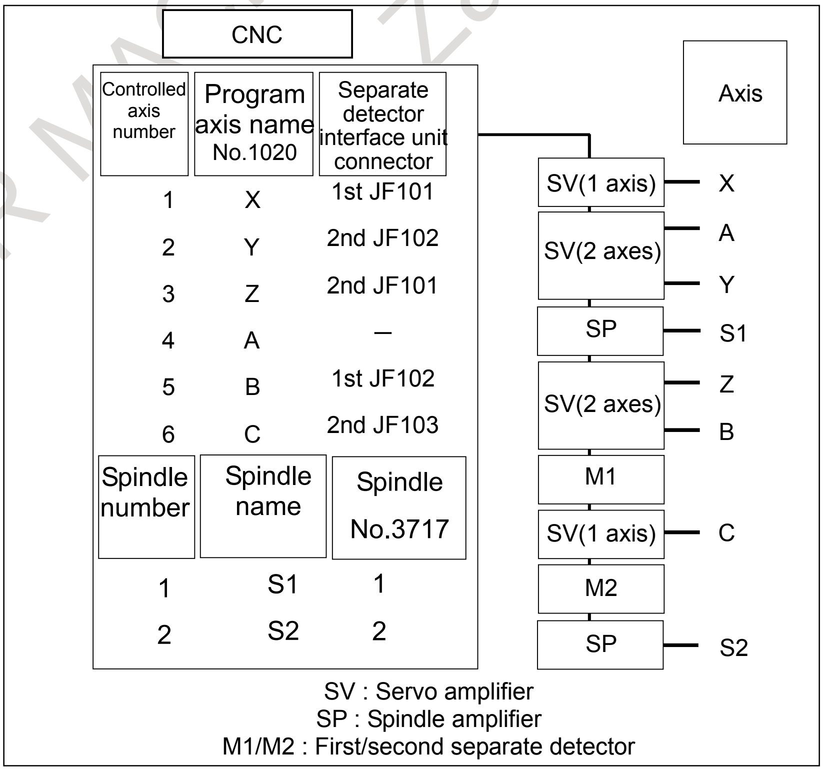


Fig. 1.4.3 (c) Setting example (separate detector interface unit)

Table 1.4.3 (b) Setting example (separate detector interface unit)

No.	1902#0 FMD
	1

No.	3717
S1	1
S2	2

No.	1023	24096	24097	24098 to 24103
X	1	1	0	-
Y	3	0	2	-
Z	4	0	1	-
A	2	0	0	-
B	5	2	0	-
C	6	0	3	-

No.	24000	24001	24002	24003	24004	24005	24006	24007
	1001	1002	1003	2001	1004	1005	3001	1006
No.	**24008**	**24009**	**24010 to 24031**			**24032 to 24063**		
	3002	2002	1000			-		

NOTE
For a parameter value indicated with a hyphen (-), be sure to set 0.

- For servo HRV2 control

When servo HRV2 control is used, specify 1 + 8n, 2 + 8n, 3 + 8n, 4 + 8n, 5 + 8n, and 6 + 8n (n = 0, 1, 2, ..., 9) like 1, 2, 3, 4, 5, 6, 9, 10, ..., 77, and 78 in parameter No. 1023 as servo axis numbers.

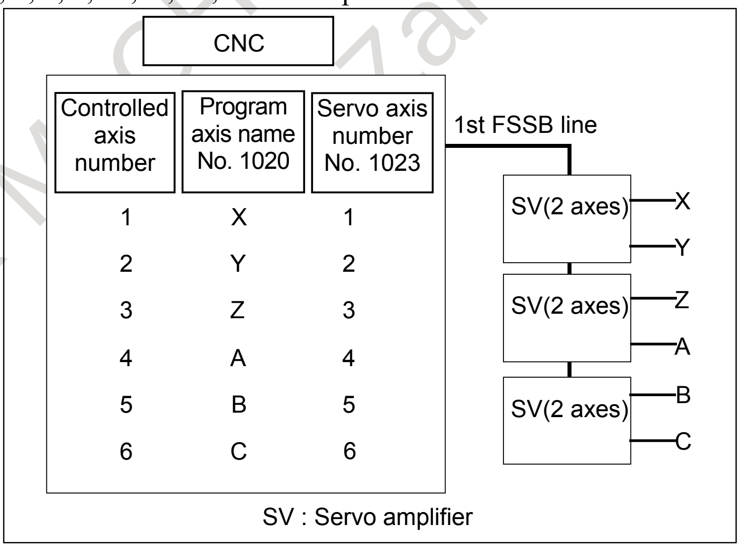


Fig. 1.4.3 (d) Setting example (servo HRV2 control)

Table 1.4.3 (c) Setting example (servo HRV2 control)

No.	1902#0 FMD
	1

No.	24000	24001	24002	24003	24004	24005
	1001	1002	1003	1004	1005	1006
No.	**24006 to 24031**					
	1000					

- For servo HRV3 control

When servo HRV3 control is used, specify 1 + 8n, 2 + 8n, 3 + 8n, 4 + 8n (n = 0, 1, 2, ..., 9) like 1, 2, 3, 4, 9, 10, ..., 75, and 76 in parameter No. 1023 as servo axis numbers.

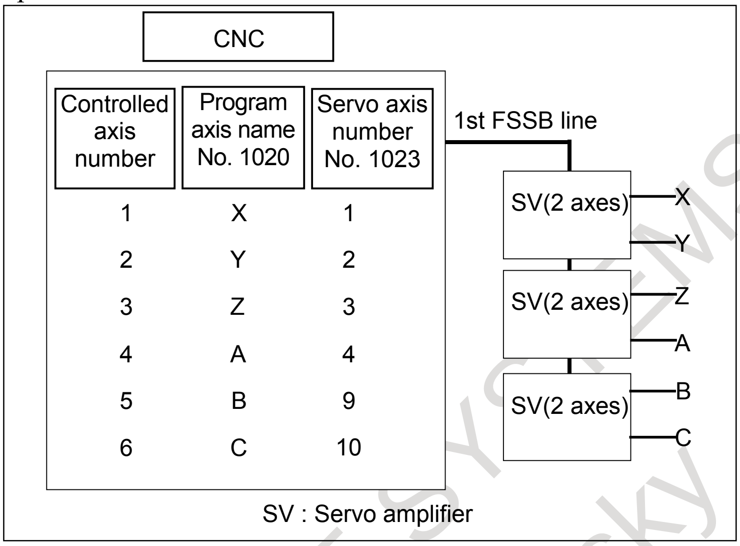


Fig. 1.4.3 (e) Setting example (servo HRV3 control)

Table 1.4.3 (d) Setting example (servo HRV3 control)

No.	1902#0 FMD
	1

No.	24000	24001	24002	24003	24004	24005
	1001	1002	1003	1004	1009	1010
No.	**24006 to 24031**					
	1000					

NOTE

When servo HRV3 control is used, the number of units connected to each FSSB line is limited. For details, refer to "Connection to the amplifier" in "Connection Manual (Hardware)" (B-64693EN).

1.4.3.1 FSSB setting screen

The FSSB setting screen displays FSSB-related information. This information can also be specified by the operator.

1 Press function key [SYSTEM].

2 To display [FSSB], press the [▷] continuous menu page key several times.

3 Pressing the soft key [FSSB] causes the [CONNECTION STATUS] screen (or the previously selected FSSB setting screen) to appear, with the following soft keys displayed.

Fig. 1.4.3.1 (a) Soft keys on the FSSB setting screen

There are seven FSSB setting screens: [CONNECTION STATUS], [SERVO AMPLIFIER SETTING], [SPINDLE AMPLIFIER SETTING], [SEPARATE DETECTOR INTERFACE UNIT], [AXIS SETTING], [SERVO AMPLIFIER MAINTENANCE], and [SPINDLE AMPLIFIER MAINTENANCE].

(1) Pressing the soft key [CONECT STATUS] causes the [CONNECTION STATUS] screen to appear.
(2) Pressing the soft key [SERVO AMP] causes the [SERVO AMPLIFIER SETTING] screen to appear.
(3) Pressing the soft key [SPNDLE AMP] causes the [SPINDLE AMPLIFIER SETTING] screen to appear.
(4) Pressing the soft key [PULSE MODULE] causes the [SEPARATE DETECTOR INTERFACE UNIT] screen to appear.
(5) Pressing the soft key [AXIS] causes the [AXIS SETTING] screen to appear.
(6) Pressing the soft key [SERVO MAINTE] causes the [SERVO AMPLIFIER MAINTENANCE] screen to appear.
(7) Pressing the soft key [SPNDLE MAINTE] causes the [SPINDLE AMPLIFIER MAINTENANCE] screen to appear.

(1) Connection status screen

The connection status screen displays the connection status of slaves connected to the FSSB at power-on.

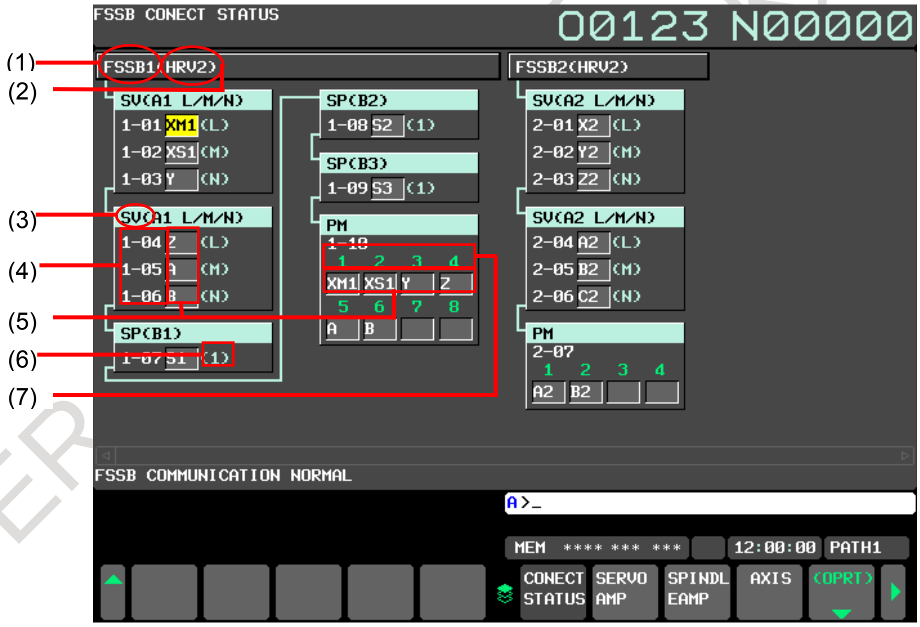


Fig. 1.4.3.1 (b) Connection status screen

The connection status screen displays the following items:

<1> FSSB1,FSSB2 ....FSSB line number
The FSSB line number is displayed. (FSSB1: First FSSB line, FSSB2: Second FSSB line)

<2> HRV2,HRV3, HRV- ....Current loop
The current loop for each FSSB line is displayed. “HRV-” may be displayed when no servo amplifier is connected to the FSSB or an FSSB-related alarm is issued.

<3> SV,SP,PM ....Slave type
The type of slave connected to the FSSB is displayed. (SV: Servo amplifier, SP: Spindle amplifier, PM: Separate detector interface unit)

<4> 1-01 to 1-32, 2-01 to 2-32 ....Slave number

An FSSB line number (1: First FSSB line, 2: Second FSSB line), a hyphen (-), and a slave number (connection number for the line) are displayed. (The maximum number of slaves per line is 32.)

<5> XM1,XS1,Y,Z,A,B .... Program axis name, Spindle name
The program axis name or spindle name set for each amplifier or separate detector interface unit is displayed.

<6> L,M,N,1 .... Amplifier axis order
The axis order for each amplifier is displayed. (L: First axis for a servo amplifier, M: Second axis for a servo amplifier, N: Third axis for a servo amplifier, 1: First spindle for a spindle amplifier)

<7> 1 to 8 .... Connector number
The connector number of a separate detector interface unit is displayed.

NOTE
When using the serial feedback dummy function by first axis of a multi-axis amplifier, the connection status is not correctly displayed.

When the softkey [(OPRT)] is pressed, the following softkey is displayed.

When the softkey [CURRENT] is pressed, current connection status is displayed.
When the alarm related to FSSB occurs and the softkey [ALARM] is pressed, the problem part is displayed in red judging from the data that compares with the FSSB connection state normally established in the past and the present FSSB connection.
When the softkey [SYSA HISTRY1] or [SYSA HISTRY2] is pressed, the problem part is displayed in red juding from the data that compares with the FSSB connection state normally established in the past and System alarm history data.

(2) Servo amplifier setting screen
The servo amplifier setting screen displays servo amplifier information.

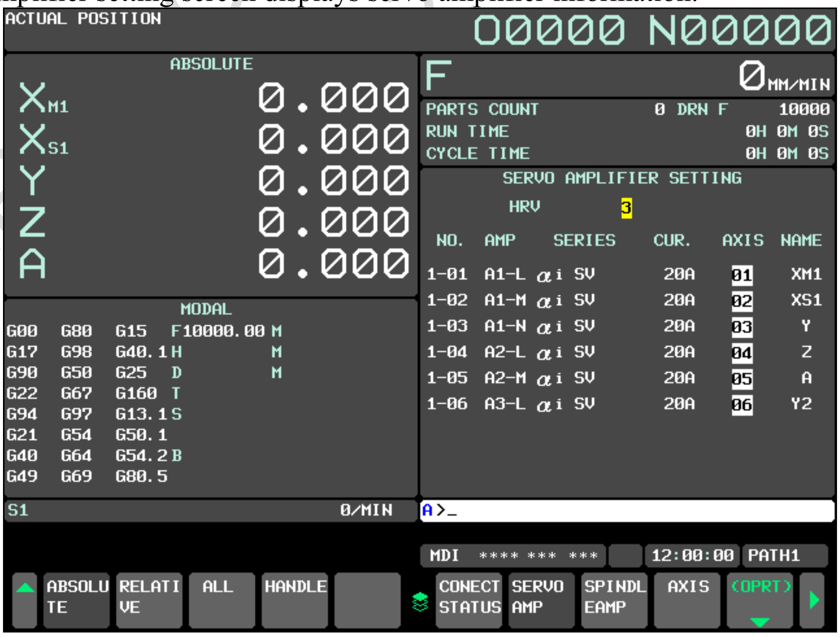


Fig. 1.4.3.1 (c) Servo amplifier setting screen

The servo amplifier setting screen consists of the following items:

- HRV .... Current loop

The current loop to be set at FSSB automatic setting is displayed. This value does not indicate the current effective current loop. (2: Servo HRV2 control, 3: Servo HRV3 control)

- NO..Slave number
  An FSSB line number (1: First FSSB line, 2: Second FSSB line), a hyphen (-), and a slave number (connection number for the line) are displayed. (The maximum number of slaves per line is 32.)
- AMP..Amplifier type
  This consists of the letter A, which stands for “servo amplifier”, a number indicating the placing of the servo amplifier, as counted from that nearest to the CNC, and an alphabetic character indicating the axis order in the servo amplifier (L: First axis, M: Second axis, N: Third axis).
- The following items are displayed as servo amplifier information:
  - SERIES.................Servo amplifier type and series
    (The display of the series name “β*i* SV” is not supported)
  - CUR. ....Maximum rating current
- AXIS ....Controlled axis number
  The controlled axis number assigned to the servo amplifier is displayed. “0” is displayed if an FSSB-related alarm is issued or no controlled axis number is assigned.
- NAME..Program axis name
  The program axis name corresponding to a particular controlled axis number set in parameter No. 1020 is displayed. When the axis number is 0, nothing is displayed.

(3) Spindle amplifier setting screen
The spindle amplifier setting screen displays spindle amplifier information.

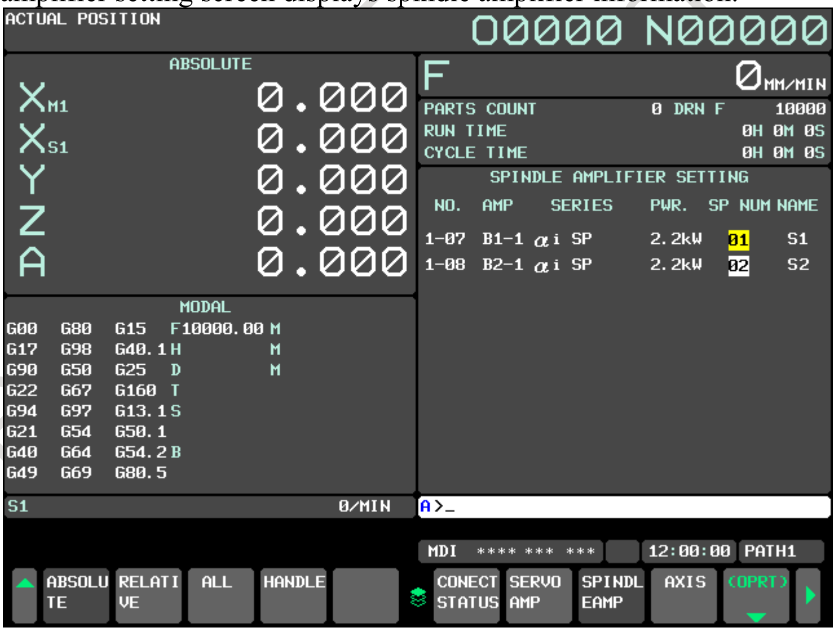


Fig. 1.4.3.1 (d) Spindle amplifier setting screen

The spindle amplifier setting screen consists of the following items:

- NO..Slave number
  An FSSB line number (1: First FSSB line, 2: Second FSSB line), a hyphen (-), and a slave number (connection number for the line) are displayed. (The maximum number of slaves per line is 32.)
- AMP..Amplifier type
  This consists of the letter B, which stands for “spindle amplifier”, a number indicating the placing of the spindle amplifier, as counted from that nearest to the CNC, and an alphabetic character indicating the axis order in the spindle amplifier (1: First spindle for a spindle amplifier).

- The following items are displayed as spindle amplifier information:
  - SERIES..................Spindle amplifier type and series
  - PWR.......................Maximum output
- SP NUM...Spindle number
  The spindle number assigned to the spindle amplifier is displayed. “0” is displayed if an FSSB-related alarm is issued or no spindle number is assigned.
- NAME...Spindle name
  The spindle name corresponding to the spindle number is displayed. When the spindle number is 0, nothing is displayed.

(4) Separate detector interface unit screen
The separate detector interface unit screen displays information on separate detector interface units.

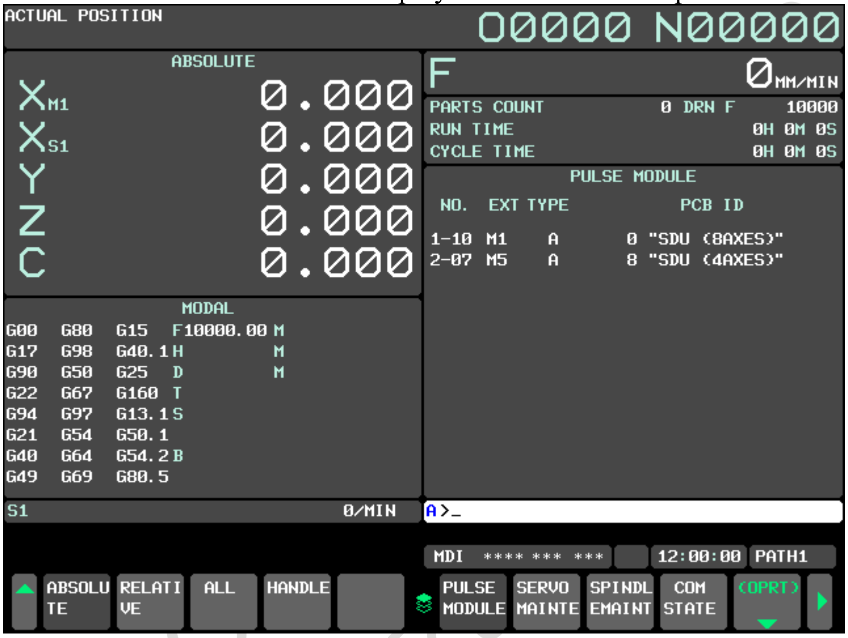


Fig. 1.4.3.1 (e) Separate detector interface unit screen

The separate detector interface unit screen displays the following items:

- NO...Slave number
  An FSSB line number (1: First FSSB line, 2: Second FSSB line), a hyphen (-), and a slave number (connection number for the line) are displayed. (While the maximum number of slaves per line is 32, the maximum number of separate detector interface units per line is 4.)
- The following items are displayed as separate detector interface unit information:
  - EXT
    This consists of the letter M, which stands for "separate detector interface unit", and a number indicating the placing of the separate detector interface unit, as counted from that nearest to the CNC. For the second FSSB line, M5 is displayed for the first separate detector interface unit since the number starts from 5.
  - TYPE
    This is a letter indicating the type of the separate detector interface unit.
  - PCB ID
    The ID of the separate detector interface unit is displayed.
    The separate detector interface unit ID is followed by SDU (8AXES) when 8-axes separate detector interface unit or SDU (4AXES) when 4-axes separate detector interface unit.

(5) Axis setting screen
The axis setting screen displays the information of axis.

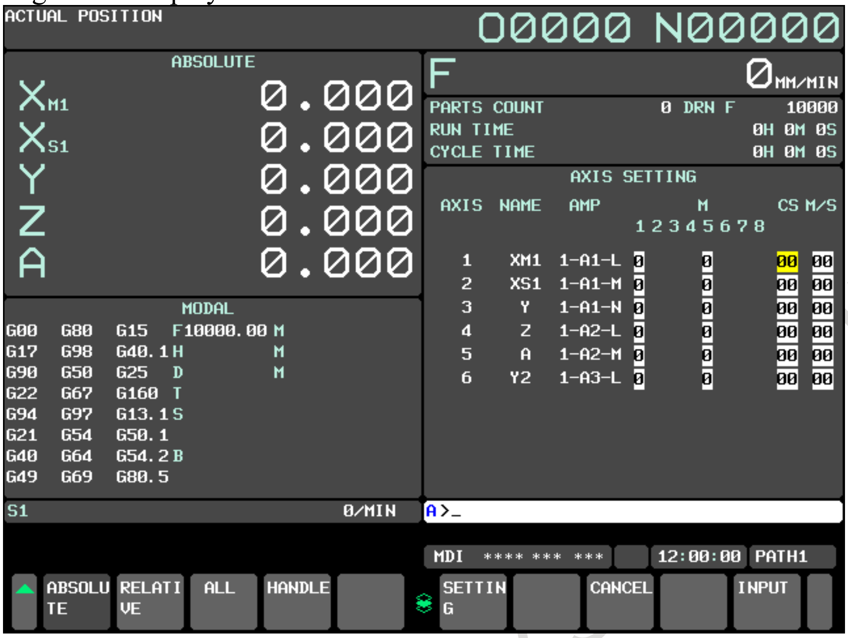


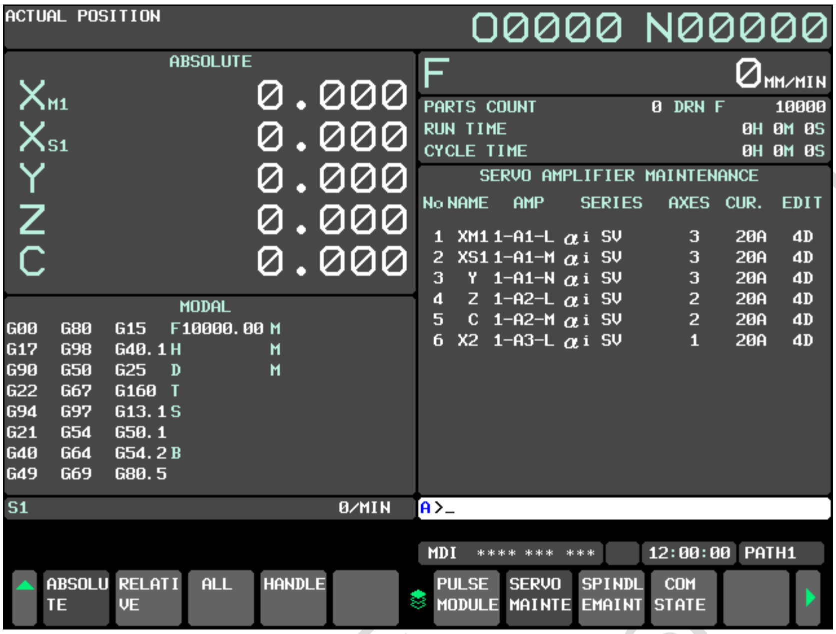
Fig. 1.4.3.1 (f) Axis setting screen

The axis setting screen displays the following items. Any item that cannot be set is not displayed. (When the first and fifth separate detector interface units are connected and Cs contour control and tandem control can be used, the screen shown in Fig. 1.4.3.1 (f) is displayed.)

- AXIS ....Controlled axis number
  This item is the placing of the NC controlled axis.
- NAME..........Program axis name for each axis
- AMP.............FSSB line number and amplifier type of the servo amplifier connected to each axis
- M1 ....Connector number of the first separate detector interface unit
- M2 ....Connector number of the second separate detector interface unit
- M3 ....Connector number of the third separate detector interface unit
- M4 ....Connector number of the fourth separate detector interface unit
- M5 ....Connector number of the fifth (first unit for the second FSSB line) separate detector interface unit
- M6 ....Connector number of the sixth separate detector interface unit
- M7 ....Connector number of the seventh separate detector interface unit
- M8 ....Connector number of the eighth separate detector interface unit
  Connector numbers set by FSSB automatic setting are displayed.
- Cs ....Cs contour controlled axis
  The spindle number for the Cs contour controlled axis set by FSSB automatic setting is displayed.
- M/S...............Master axis / Slave axis (Slave axis / Dummy axis)
  Either of the following settings is displayed: Master axis/slave axis setting for tandem control or slave axis/dummy axis setting for the electronic gear box (EGB) set by FSSB automatic setting.

The M1 to M8, Cs, and M/S values are to be set by FSSB automatic setting and do not indicate current effective settings. The previous values set normally are displayed first after power-on. “0” is displayed when an FSSB-related alarm is issued.

(6) Servo amplifier maintenance screen
The servo amplifier maintenance screen displays maintenance information for servo amplifiers. This screen consists of the following two pages, either of which can be selected by pressing the cursor keys [←] and [→].



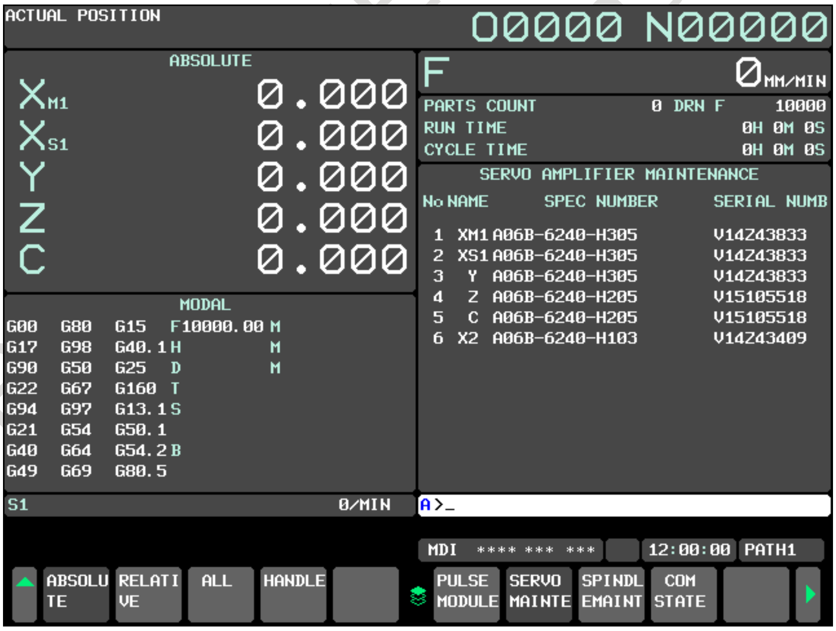
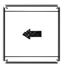
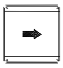


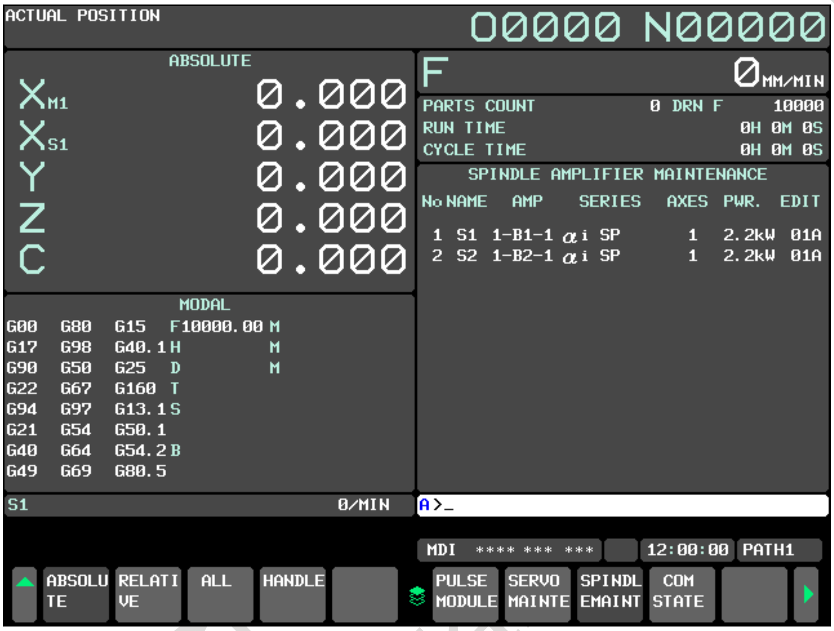
Fig. 1.4.3.1 (g) Servo amplifier maintenance screen

The servo amplifier maintenance screen displays the following items:

- No........................ Controlled axis number
- NAME................. Program axis name for each axis
- AMP.................... FSSB line number and amplifier type of the servo amplifier connected to each axis
- SERIES .... Type and series of the servo amplifier connected to each axis (The display of the series name "β*i*SV" is not supported)
- AXES .... Maximum number of axes controlled by a servo amplifier connected to each axis

- CUR. .... Maximum rating current for servo amplifiers connected to each axis
- EDIT .... Version number of a servo amplifier connected to each axis
- SPEC NUMBER. Amplifier drawing number of the servo amplifier connected to each axis
- SERIAL NUMB . Serial number of the servo amplifier connected to each axis

(7) Spindle amplifier maintenance screen
The spindle amplifier maintenance screen displays maintenance information for spindle amplifiers. This screen consists of the following two pages, either of which can be selected by pressing the cursor keys  and .



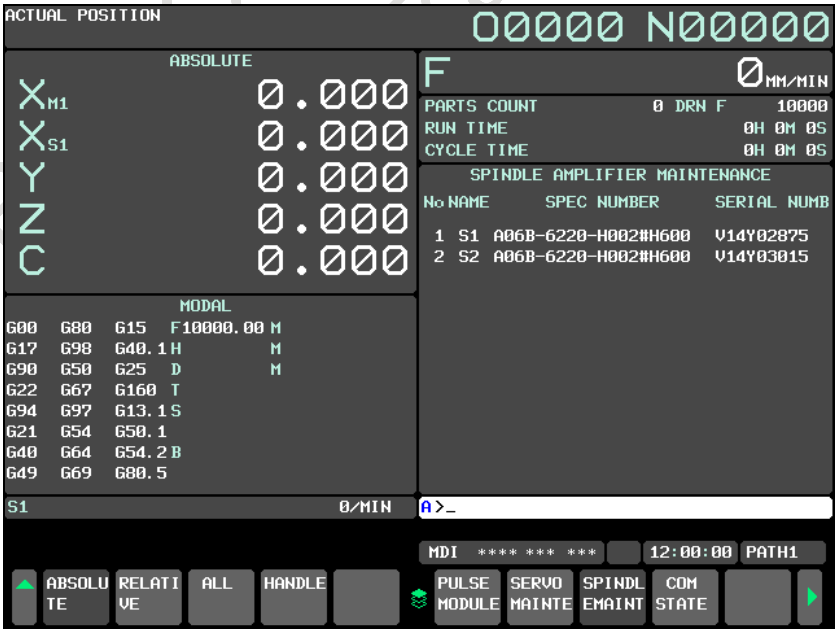


Fig. 1.4.3.1 (h) Spindle amplifier maintenance screen

The spindle amplifier maintenance screen displays the following items:

- No........................ Spindle number
- NAME................. Spindle name

- AMP .... FSSB line number and amplifier type of the spindle amplifier connected to each axis
- SERIES .... Type and series of the spindle amplifier connected to each axis
- AXES .... Maximum number of axes controlled by a spindle amplifier connected to each axis
- PWR.................... Rated output of the spindle amplifier connected to each axis
- EDIT .... Version number of a spindle amplifier connected to each axis
- SPEC NUMBER. Amplifier drawing number of the spindle amplifier connected to each axis
- SERIAL NUMB . Serial number of the spindle amplifier connected to each axis

1.4.3.2 FSSB automatic setting procedure

To perform FSSB automatic setting, set items on the FSSB setting screens in (1) to (3) below.

(1) Servo amplifier setting screen

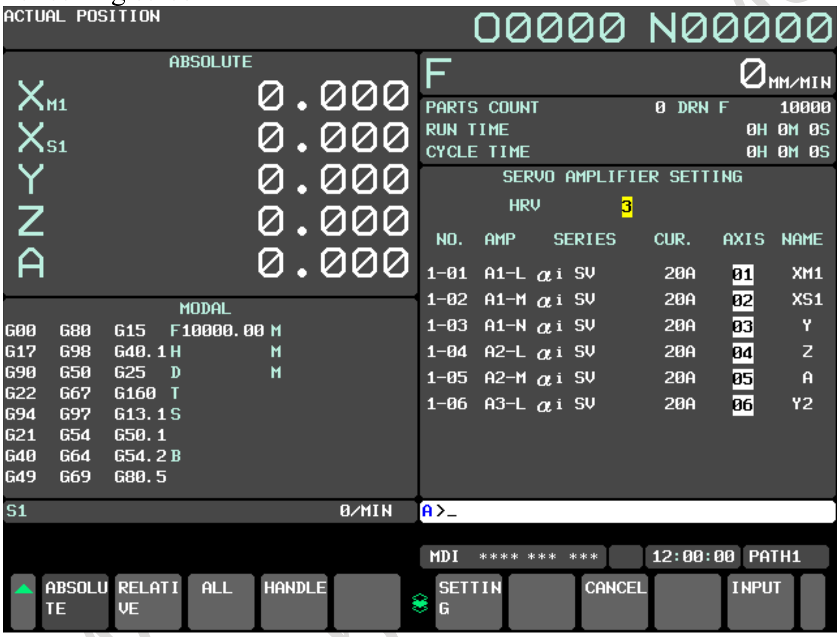


Fig. 1.4.3.2 (a) Servo amplifier setting

The servo amplifier setting screen displays the following items:

- HRV .... Current loop
  For this item, enter a value between 2 to 3.
  If a number that falls outside this range is entered, the warning message, “DATA IS OUT OF RANGE” appears.
- AXIS .... Controlled axis number
  For this item, enter a value of between 0 and the maximum number of controlled axes.
  If a number that falls outside this range is entered, the warning message, “DATA IS OUT OF RANGE” appears. Setting 0 means that the relevant servo amplifier is not used.

(2) Spindle amplifier setting screen

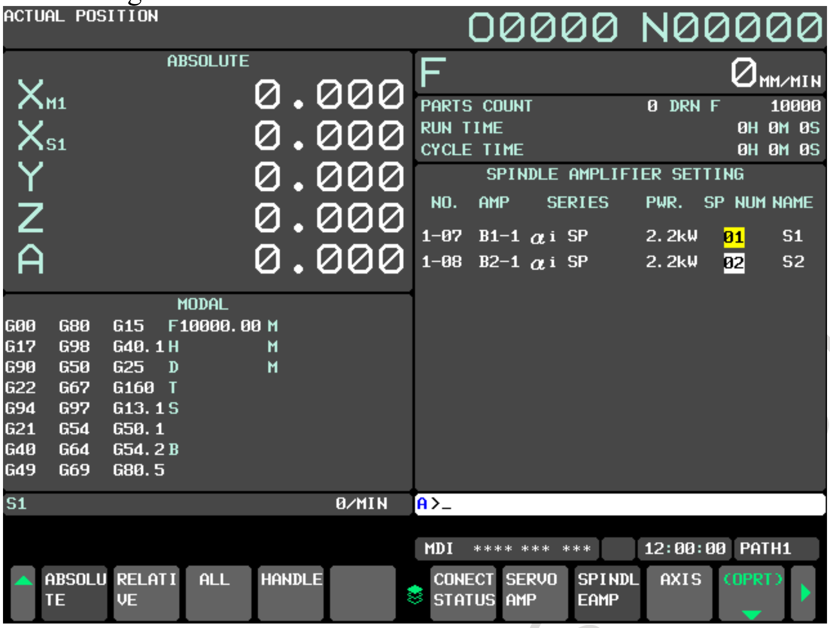


Fig. 1.4.3.2 (b) Spindle amplifier setting

The spindle amplifier setting screen displays the following items:

- SP NUM.............. Spindle number
  For this item, enter a value of between 0 and the maximum number of spindles.
  If a number that falls outside this range is entered, the warning message, "DATA IS OUT OF RANGE" appears. Setting 0 means that the relevant spindle amplifier is not used.

(3) Axis setting screen

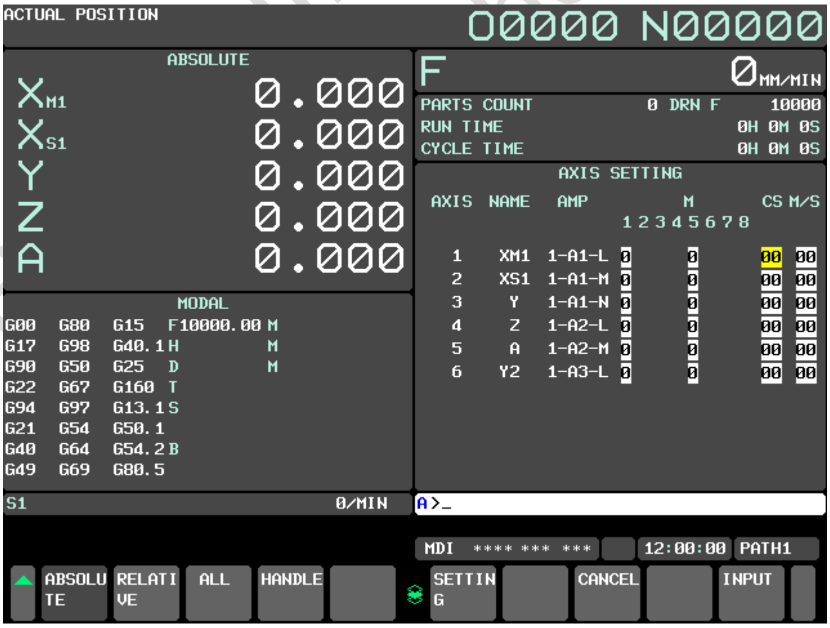


Fig. 1.4.3.2 (c) Axis setting

On the axis setting screen, the following items can be specified:

- M1............. Connector number of the first separate detector interface unit
- M2............. Connector number of the second separate detector interface unit
- M3............. Connector number of the third separate detector interface unit
- M4............. Connector number of the fourth separate detector interface unit

- M5............. Connector number of the fifth (first unit for the second FSSB line) separate detector interface unit
- M6............. Connector number of the sixth separate detector interface unit
- M7............. Connector number of the seventh separate detector interface unit
- M8............. Connector number of the eighth separate detector interface unit
  For an axis that uses each separate detector interface unit, enter a connector number using a number 1 to 8 (maximum number of connectors on a separate detector interface unit).
  When a separate detector interface unit is not used, enter 0. If a number that falls outside this range is entered, the warning message, "DATA IS OUT OF RANGE" appears. For a separate detector interface unit which is not connected, items are not displayed and values cannot be entered.

Table 1.4.3.2 (a) Connectors and corresponding connector numbers

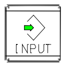
Connector	Connector number
JF101	1
JF102	2
JF103	3
JF104	4
JF105	5
JF106	6
JF107	7
JF108	8

- CS.............. Cs contour controlled axis
  Enter a spindle number between 1 and the maximum number of spindles for the Cs contour controlled axis. When a Cs contour controlled axis is not used, enter 0. If a number that falls outside this range is entered, the warning message, "DATA IS OUT OF RANGE" appears. When Cs contour control cannot be used, this item is not displayed and any value cannot be entered.
- M/S............ Master axis / Slave axis (Slave axis / Dummy axis)
  Enter an odd number for the master axis and an even number for the slave axis for tandem control. These numbers must be consecutive and within a range between 1 and the maximum number of controlled axes. Enter an odd number for the slave axis and an even number for the dummy axis for the electronic gear box (EGB). These numbers must be consecutive and within a range between 1 and the maximum number of controlled axes. If a number that falls outside this range is entered, the warning message, "DATA IS OUT OF RANGE" appears. For the slave and dummy axes for EGB, set bit 0 (SYN) of parameter No. 2011 to 1. When both tandem control and electronic gear box cannot be used, this item is not displayed and any value cannot be entered.

On an FSSB setting screen (other than the connection status screen, servo amplifier maintenance screen, or spindle amplifier maintenance screen), pressing the soft key [(OPRT)] displays the following soft keys:

Fig. 1.4.3.2 (d) FSSB automatic setting soft keys

To enter data, place the machine in the MDI mode or the emergency stop state, position the cursor to a desired input item position, then enter desired data and press the soft key [INPUT].

(Alternatively, press the  key on the MDI unit.)

When the soft key [SETTING] is pressed after data has been entered, a warning message listed below is displayed if the entered data contains an error. When the data is valid, the corresponding FSSB-related parameters are set up. To restore the previous value normally set if, for example, an entered value is incorrect, press the soft key [CANCEL].
When this screen is first displayed after power-on, the previous values set normally are displayed.

FSSB automatic setting warning messages

If an invalid setting is detected at FSSB automatic setting, a warning message listed below is displayed. EGB dummy axis setting means setting an even number for M/S on the axis setting screen for an axis for which bit 0 (SYN) of parameter No. 2011 is set to 1. EGB slave axis setting means setting an odd number for M/S on the axis setting screen for an axis for which bit 0 (SYN) of parameter No. 2011 is set to 1.

Warning message	Cause
Cs and M/S are set with the same axis	On the axis setting screen, a value is specified for Cs and M/S for an axis. Do not specify any value for Cs and M/S simultaneously.
Cs and M1-8 are set with the same axis	On the axis setting screen, a value is specified for Cs and M1-8 for an axis. Do not specify any value for Cs and M1-8 simultaneously.
Same number is set in AXIS	On the servo amplifier setting screen, an axis number is set more than once. Specify each axis number only once.
Same number is set in SP NUM	On the spindle amplifier setting screen, a spindle number is set more than once. Specify each spindle number only once.
Same number is set in CS	On the axis setting screen, a value is set for Cs more than once. Specify each value for Cs only once.
Same number is set in M/S	On the axis setting screen, a value is set for M/S more than once. Specify each value for M/S only once.
AXIS and Cs are set with the same axis	An axis number for which a value is set for Cs on the axis setting screen is set for AXIS on the servo amplifier setting screen. Do not set any axis number for which a value is set for Cs, on the servo amplifier setting screen.
Too many slaves (HRV3)	The maximum number (15) of slaves per FSSB line for servo HRV3 control is exceeded. Reduce the number of slaves connected to an FSSB line to 15 or less.
Too many slaves (HRV2)	The maximum number (32) of slaves per FSSB line for servo HRV2 control is exceeded. Reduce the number of slaves connected to an FSSB line to 32 or less.
AXIS is set with EGB dummy axis	An axis number set for EGB dummy axis setting is set for AXIS on the servo amplifier setting screen. Do not set any axis number for EGB dummy axis setting, on the servo amplifier screen.
M/S setting is illegal(EGB)	For M/S, the EGB slave axis setting corresponding to an EGB dummy axis setting is not made. Make the EGB slave axis setting.
AXIS is not set with EGB slave axis	An axis number for EGB slave axis setting is not set for AXIS on the servo amplifier setting screen. Set the axis number for EGB slave axis setting, on the servo amplifier setting screen.
AXIS is not set with M/S axis	An axis number set for M/S on the axis setting screen is not set for AXIS on the servo amplifier setting screen. Set the axis number for M/S, on the servo amplifier setting screen.
EGB dummy axis setting is illegal	EGB dummy axis setting is made when 32 slaves are connected to the second FSSB line. Decrease the number of slaves connected to the second FSSB line.
M/S setting is illegal	Invalid M/S setting. Correct the M/S setting.
Setting is illegal(servo)	Invalid servo axis setting (servo amplifier setting, axis setting). Correct the servo axis setting.
Setting is illegal(spindle)	Invalid spindle setting. Correct the spindle setting.

> ⚠ **CAUTION**
> For the parameters to be specified on the FSSB setting screen, do not attempt to directly enter values on the parameter screen using the MDI or a G10 command. Use only the FSSB setting screen to enter values for these parameters.

Examples of FSSB automatic setting

Examples of FSSB automatic setting for the listed functions are shown below:

- Example 1 Servo HRV2 control (Servo HRV3 control)
- Example 2 Separate detector interface unit

- Example 3 Cs contour control
- Example 4 Tandem control
- Example 5 Electronic gear box

- Example 1 Servo HRV2 control

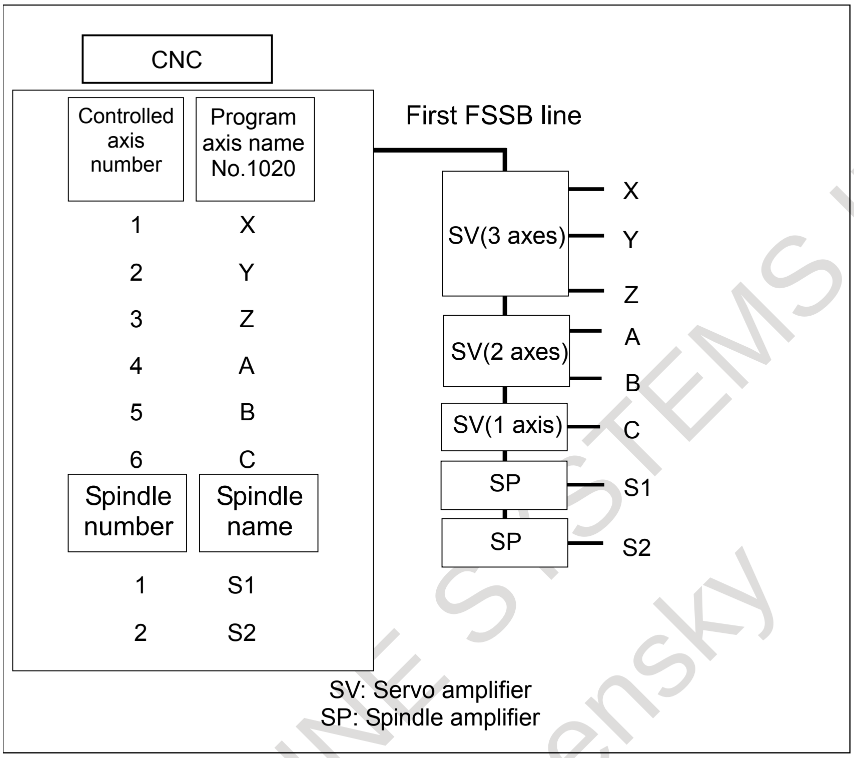
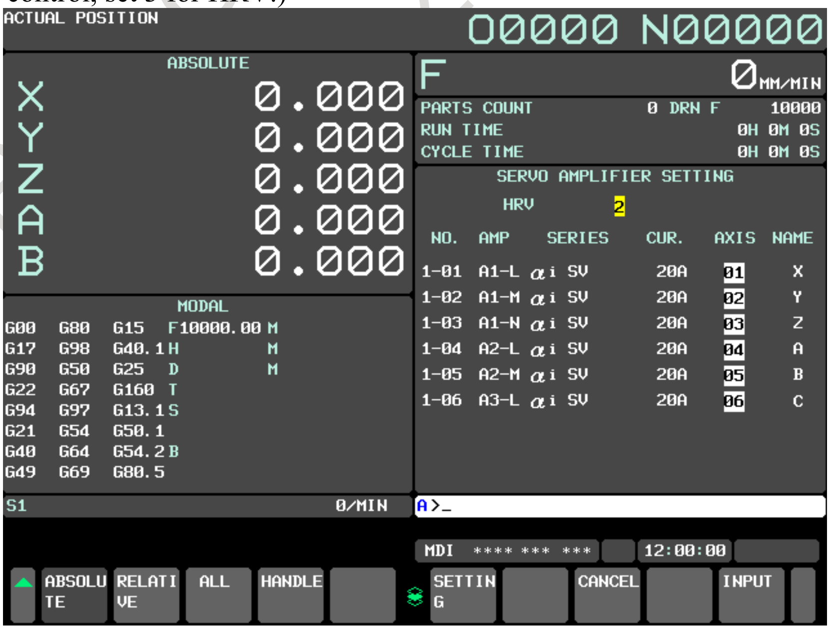
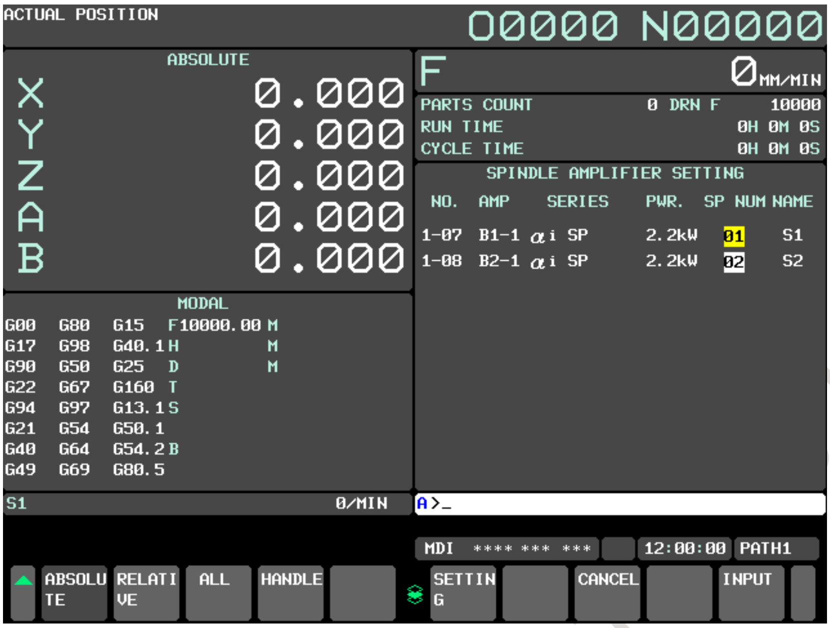


Fig. 1.4.3.2 (e) Setting example (servo HRV2 control)

<1> On the servo amplifier setting screen, enter 2 for HRV and 1, 2, 3, 4, 5, and 6 for AXIS. (To set servo HRV3 control, set 3 for HRV.)



<2> On the spindle amplifier setting screen, enter 1 and 2 for SP NUM.



<3> Press the soft key [SETTING] to perform FSSB automatic setting.

- Example 2 Separate detector interface unit

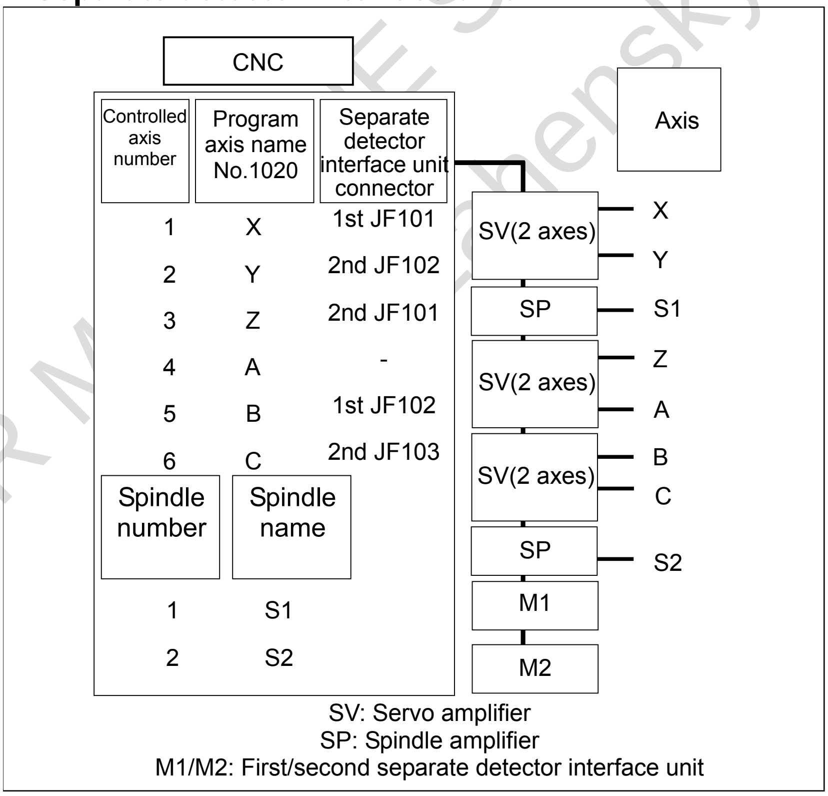
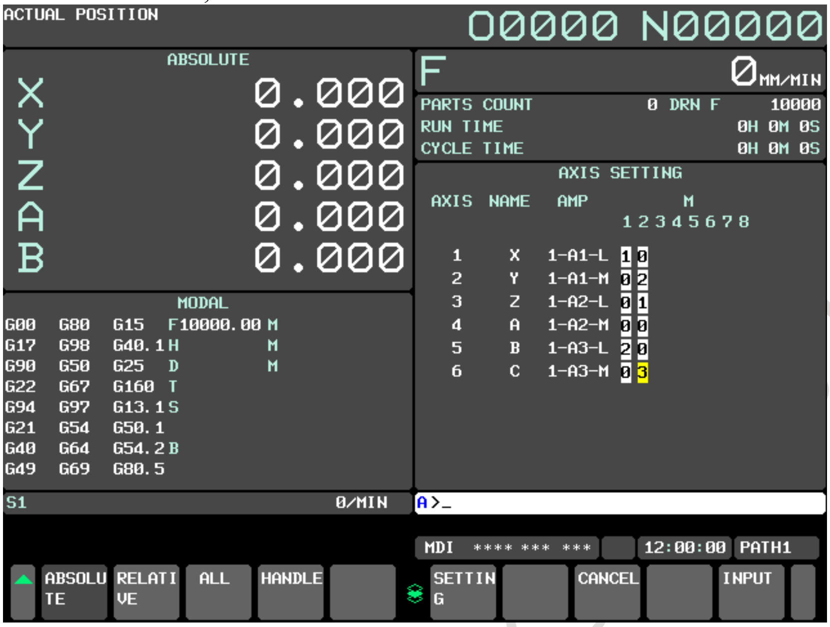


Fig. 1.4.3.2 (f) Setting example (separate detector interface unit)

<1> On the axis setting screen, enter 1 in M1 for the X-axis, 2 in M2 for the Y-axis, 1 in M2 for the Z-axis, 2 in M1 for the B-axis, and 3 in M2 for the C-axis.



<2> Press the soft key [SETTING] to perform FSSB automatic setting.

- Example 3 Cs contour control

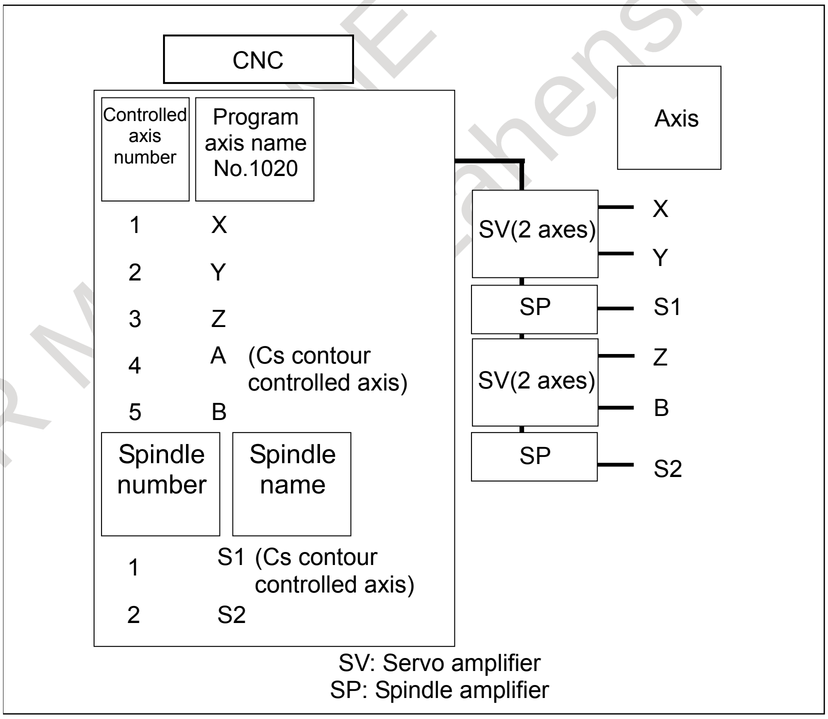
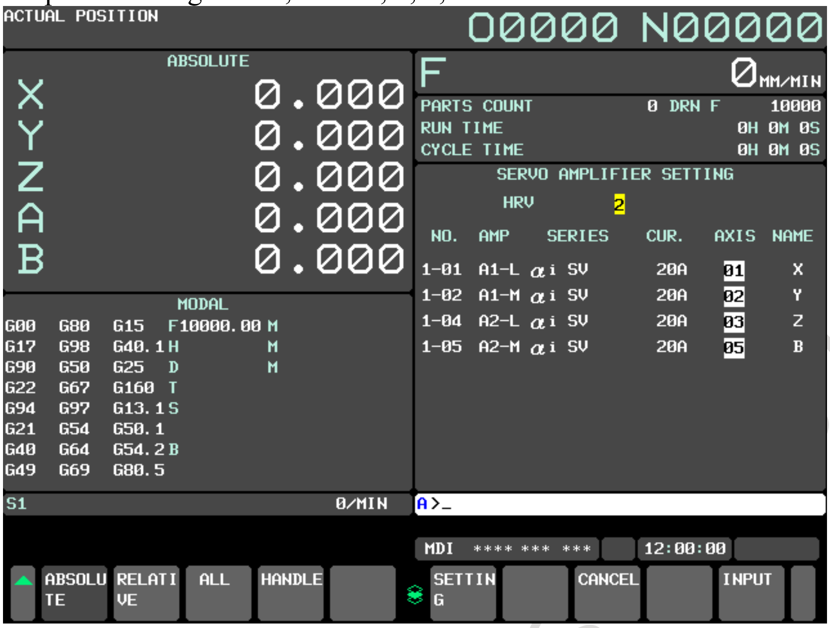
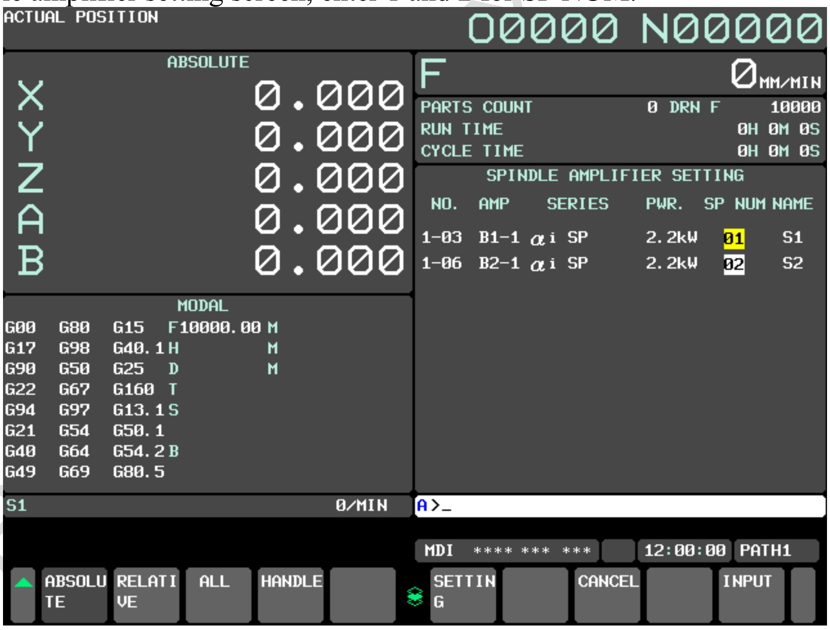


Fig. 1.4.3.2 (g) Setting example (Cs contour control)

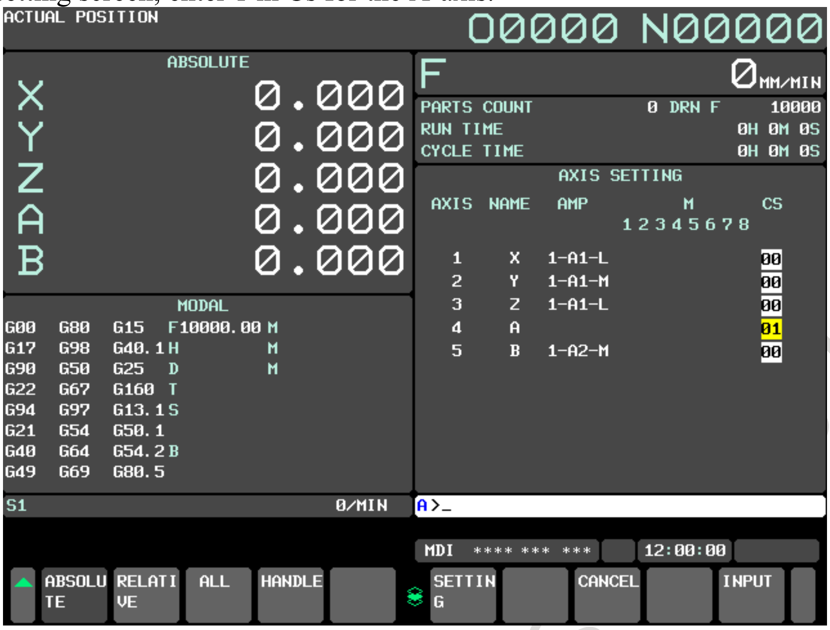
<1> On the servo amplifier setting screen, enter 1, 2, 3, and 5 for AXIS.



<2> On the spindle amplifier setting screen, enter 1 and 2 for SP NUM.



<3> On the axis setting screen, enter 1 in Cs for the A-axis.



<4> Press the soft key [SETTING] to perform FSSB automatic setting.

- Example 4 Tandem control

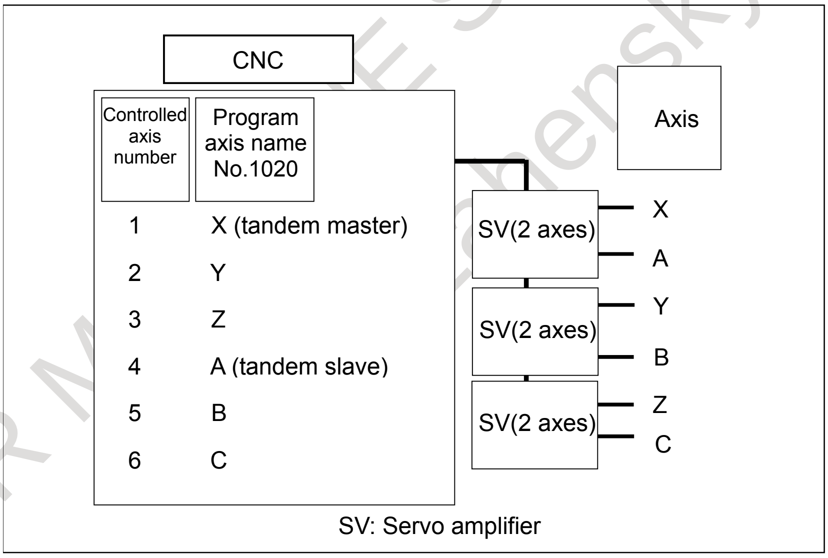
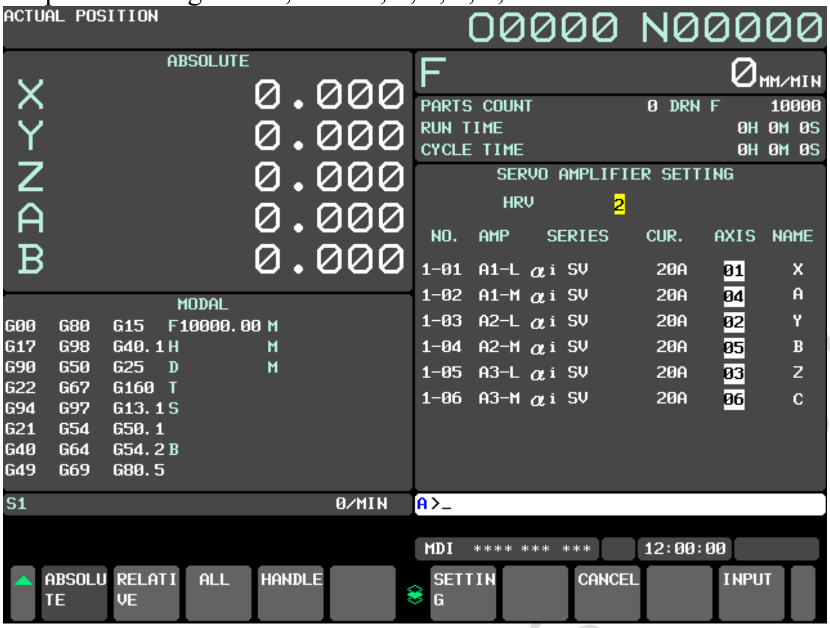
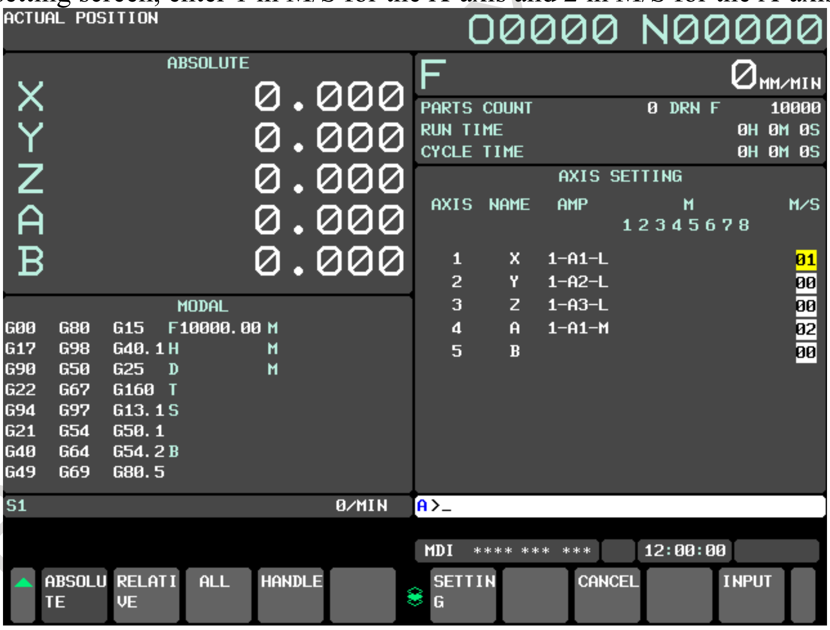


Fig. 1.4.3.2 (h) Setting example (tandem control)

<1> On the servo amplifier setting screen, enter 1, 4, 2, 5, 3, and 6 for AXIS.



<2> On the axis setting screen, enter 1 in M/S for the X-axis and 2 in M/S for the A-axis.



<3> Press the soft key [SETTING] to perform FSSB automatic setting.

- Example 5 Electronic gear box

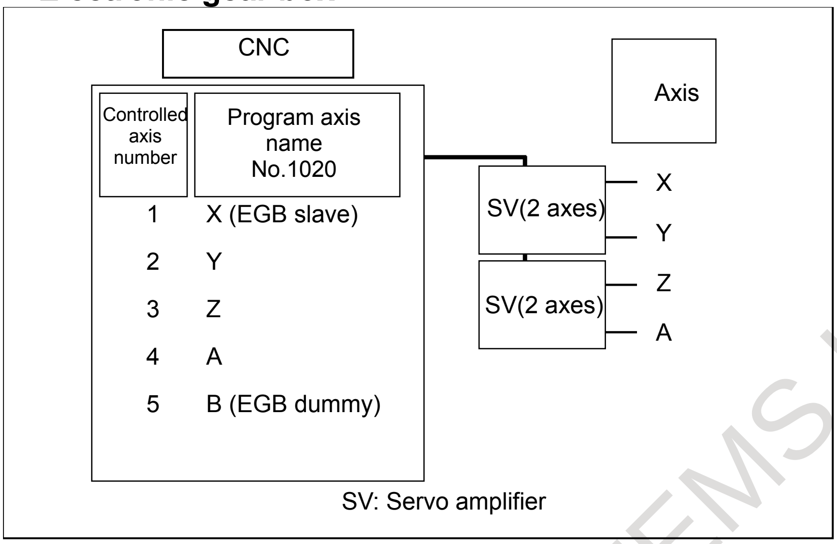
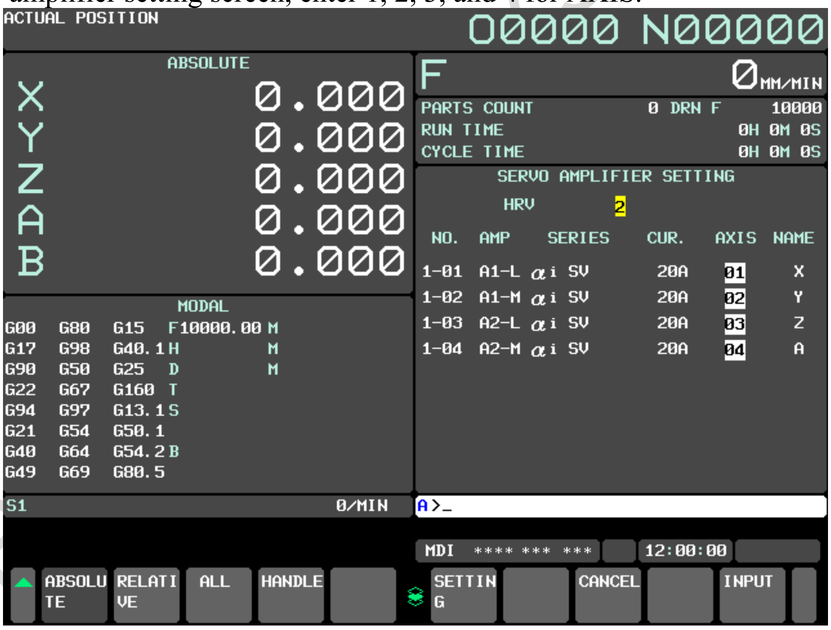
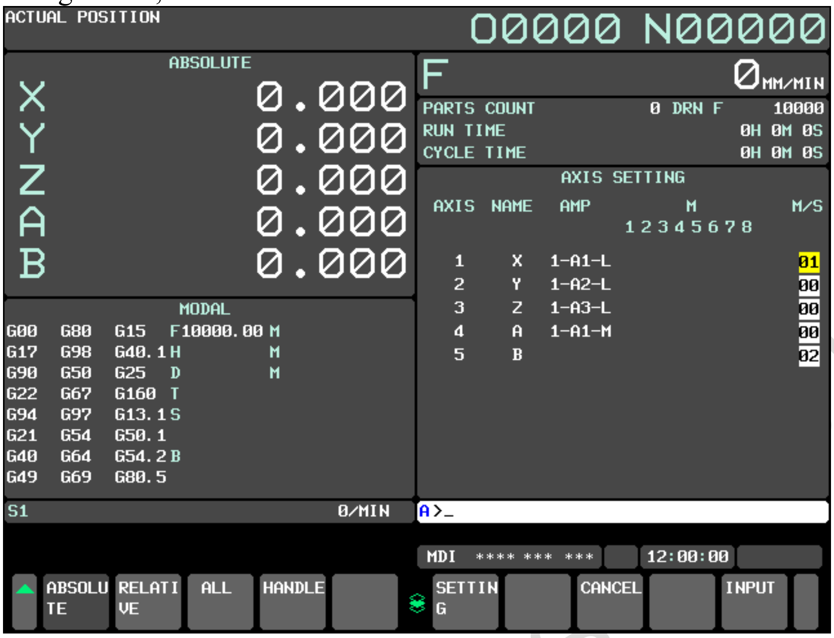


Fig. 1.4.3.2 (i) Setting example (electronic gear box)

<1> For the X- and B-axes, set bit 0 (SYN) of parameter No. 2011 to 1.
<2> On the servo amplifier setting screen, enter 1, 2, 3, and 4 for AXIS.



<3> On the axis setting screen, enter 1 in M/S for the X-axis and 2 in M/S for the B-axis.



<4> Press the soft key [SETTING] to perform FSSB automatic setting.

Parameter

1023	Number of the servo axis for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 78

This parameter associates each control axis with a specific servo axis. Usually, set the same number as a controlled axis number and its corresponding servo axis number.
The control axis number is the order number that is used for setting the axis-type parameters or axis-type machine signals
Any multiple of 8 and the value obtained by subtracting 1 from a multiple of 8 cannot be set, however. When the number of controlled axes is 7 or more, set numbers with skipping any multiple of 8 and the value obtained by subtracting 1 from a multiple of 8.

- Example of setting when the number of controlled axes is 6 or less

Parameter No.1023	Setting value
(1st axis)	1
(2nd axis)	2
(2rd axis)	3
(4th axis)	4
(5th axis)	5
(6th axis)	6

- Example of setting when the number of controlled axes is 7 or more

Parameter No.1023	Setting value
(1st axis)	1
(2nd axis)	2
(2rd axis)	3
(4th axis)	4
(5th axis)	5
(6th axis)	6
(7th axis)	9
(8th axis)	10
(9th axis)	11
(10th axis)	12
(11th axis)	13
(12th axis)	14
(13th axis)	17
(14th axis)	18

- With an axis for which Cs contour control/spindle positioning is to be performed, set -(spindle number) as the servo axis number.
  Example)
  When exercising Cs contour control on the fourth controlled axis by using the first spindle, set -1.
- For tandem controlled axes or electronic gear box (EGB) controlled axes, two axes need to be specified as one pair. So, make a setting as described below.
  Tandem axis: For a master axis, set an odd (1, 3, 5, ...) servo axis number. For a slave axis to be paired, set a value obtained by adding 1 to the value set for the master axis.
  EGB axis: For a slave axis, set an odd (1, 3, 5, ...) servo axis number. For a dummy axis to be paired, set a value obtained by adding 1 to the value set for the slave axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1902							ASE	FMD

[Input type] Parameter input
[Data type] Bit

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 FMD The FSSB setting mode is:
0: Automatic setting mode.
(When the relationship between an axis and amplifier is defined on the FSSB setting screen, parameters Nos. 1023, 2013#0, 2014#0, 3717, 11802#4, 24000 to 24103 are automatically set.
1: Manual setting 2 mode.
(Parameters Nos. 1023, 2013#0, 2014#0, 3717, 11802#4, 24000 to 24103 are to be manually set.)

#1 ASE When automatic setting mode is selected for FSSB setting (when the bit 0 (FMD) parameter No. 1902 is set to 0), automatic setting is:
0: Not completed.
1: Completed.

This bit is automatically set to 1 upon the completion of automatic setting.

	#7	#6	#5	#4	#3	#2	#1	#0
3716								A/Ss

[Input type] Parameter input
[Data type] Bit spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 A/Ss Spindle motor type is :
0: Analog spindle.
1: Serial spindle.

> **NOTE**
> 1 When FSSB setting is used, the setting of serial spindle is required.
> 2 When a serial spindle is used, enable the spindle serial output (bit 5 (SSN) of parameter No.8133 is 0).

3717	Spindle amplifier number of each spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled axes

Set a spindle amplifier number to be assigned to each spindle.
0: No spindle amplifier is connected.
1: Spindle motor connected to amplifier number 1 is used.
2: Spindle motor connected to amplifier number 2 is used.
to
n: Spindle motor connected to amplifier number n is used.

NOTE

1 In addition, it is necessary to set FSSB. Refer to Parameter Nos.24000 to 24095 for details.
2 If value of this parameter is larger than the maximum value, the alarm SP1996, "ILLEGAL SPINDLE PARAMETER SETTING" occurs.
3 If the spindle motor is treated as hypothetical Cs axis control or spindle control with servo motor, set this parameter to “0”.
4 In the display order of the parameter No.982, the spindle axes since the spindle axis that 0 is set to this parameter become invalid, too. However, the case that the spindle motor is treated as hypothetical Cs axis control or spindle control with servo motor is excepted.
Example) On the following setting, S3 and S4 spindle axes are invalid when the spindle motor of S3 is not treated as hypothetical Cs axis control or spindle control with servo motor.

Display order of No.982	Setting value of No.982	Setting value of No.3717
S1	1	1
S2	2	2
S3	1	0
S4	2	3

	#7	#6	#5	#4	#3	#2	#1	#0
14476			SSC					

[Input type] Parameter input
[Data type] Bit

NOTE

When this parameter is set, the power must be turned off before operation is continued.

#5 SSC The number of ATR values for a separate detector interface unit connector is:
0: Only 1.
1: More than 1.

NOTE

When this parameter is set to 1, FSSB automatic setting cannot be performed.

24000	ATR value corresponding to slave 01 on first FSSB line
24001	ATR value corresponding to slave 02 on first FSSB line
to	to
24031	ATR value corresponding to slave 32 on first FSSB line

> **NOTE**
> When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1001 to 1046, 2001 to 2016, 3001 to 3004, 1000

Each of these parameters sets the value (ATR value) of the address translation table corresponding to each of slave 1 to slave 32 on first FSSB line (first optical connector).

The slave is a generic term for servo amplifiers, spindle amplifiers and separate detector interface units connected via an FSSB optical cable to the CNC. Numbers 1 to 32 are assigned to slaves, with younger numbers sequentially assigned to slaves closer to the CNC.

A 2-axis amplifier consists of two slaves, and a 3-axis amplifier consists of three slaves. In each of these parameters, set a value as described below, depending on whether the slave is an amplifier, separate detector interface unit, or nonexistent.

- When the slave is a servo amplifier:
  Set the axis number of a servo amplifier to allocate (value set with parameter No. 1023) plus 1000.
- When the slave is a spindle amplifier:
  Set the spindle number of a spindle to allocate (value set with parameter No. 3717) plus 2000.
- When the slave is a separate detector interface unit:
  Set 3001, 3002, 3003, and 3004, respectively, for the first (one connected nearest to the CNC), second, third, and fourth separate detector interface units.
- When the slave is nonexistent:
  Set 1000.

> **NOTE**
> 1 When Electronic gear box (EGB) function is used
> Although an amplifier is not actually required for an EGB dummy axis, set this parameter with assuming that a dummy amplifier is connected. To put it another way, specify this parameter with a value set in the EGB dummy axis parameter No. 1023 plus 1000, instead of “1000”, as an address translation table value for one of non-existent slaves.
> 2 When the FSSB is set to the automatic setting mode (when the bit 0 (FMD) of parameter No. 1902 is set to 0), parameter Nos. 24000 to 24031 are automatically set as data is input on the FSSB setting screen. When the manual setting 2 mode is set (when the bit 0 (FMD) of parameter No. 1902 is set to 1), be sure to directly set values in parameter Nos. 24000 to 24031.

Example of axis configuration and parameter settings

- Example 1 Typical setting

CNC

Controlled axis number	Program axis name No.1020	Servo axis No.1023
1	X	1
2	Y	3
3	Z	4
4	A	2
5	B	5
6	C	6

Spindle number	Spindle amplifier number No.3717
1(S1)	1
2(S2)	2

Amplifier	Slave number	ATR No.24000 -24031	Axis
SV(1 axis)	1	1001	X
SV(2 axes)	2	1002	A
	3	1003	Y
SV(2 axes)	4	1004	Z
	5	1005	B
SP	6	2001	S1
M1	7	3001	(M1)
SV(1 axis)	8	1006	C
M2	9	3002	(M2)
SP	10	2002	S2
	11 to 32	1000	(None)

CNC

Controlled axis number	Program axis name No.1020	Servo axis No.1023
1	X	1
2	Y	3
3	Z	4
4	A	2
5	B	5
6	C	6

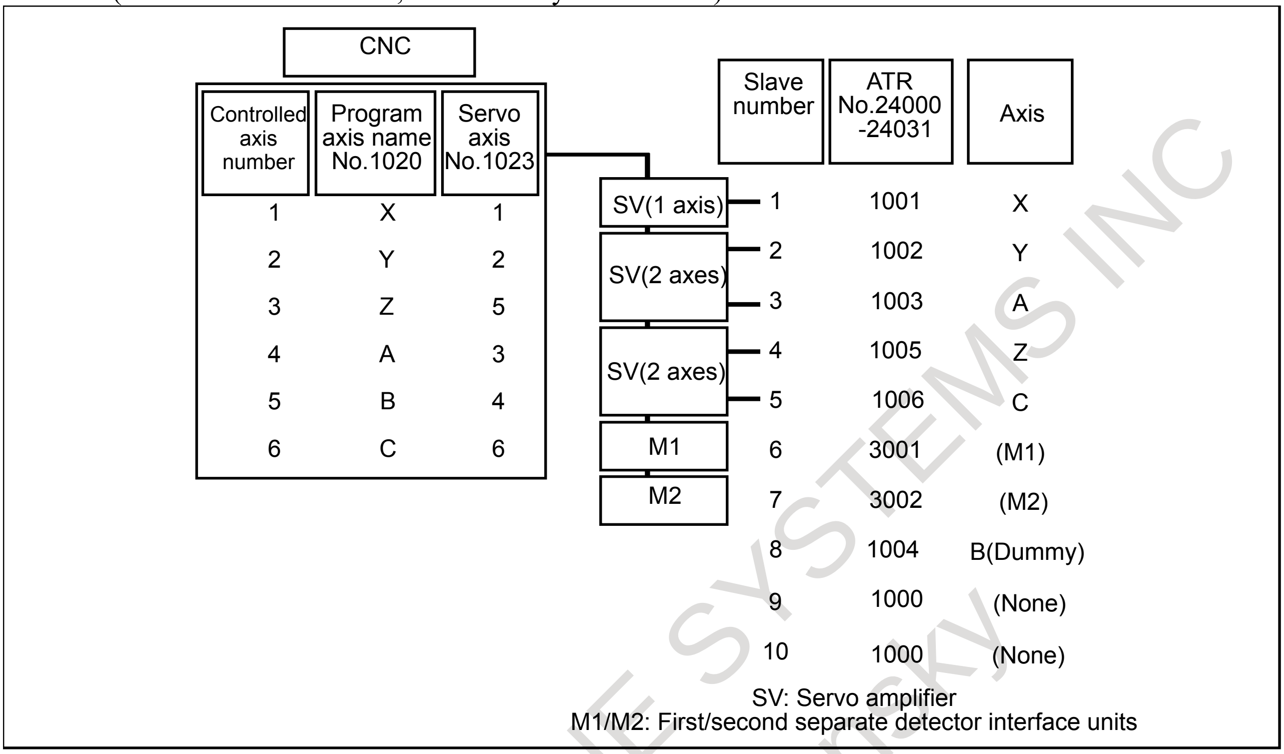
Spindle number	Spindle amplifier number No.3717
1(S1)	1
2(S2)	2

Amplifier	Slave number	ATR No.24000 -24031	Axis
SV(1 axis)	1	1001	X
SV(2 axes)	2	1003	Y
	3	1004	Z
SV(2 axes)	4	1002	A
	5	1005	B
SP	6	2002	S2
M1	7	3001	(M1)
SV(1 axis)	8	1006	C
M2	9	3002	(M2)
SP	10	2001	S1
	11 to 32	1000	(None)

SV: Servo amplifier
SP: Spindle amplifier
M1/M2: First/second separate detector interface units

- Example 2 Setting with a dummy axis in use

Example of axis configuration and parameter settings when the electronic gear box (EGB) function is used
(EGB slave axis: A-axis, EGB dummy axis: B-axis)



24032	ATR value corresponding to slave 01 on second FSSB line
24033	ATR value corresponding to slave 02 on second FSSB line
to	to
24063	ATR value corresponding to slave 32 on second FSSB line

> **NOTE**
> When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1001 to 1046, 2001 to 2016, 3005 to 3008, 1000

Each of these parameters sets the value (ATR value) of the address translation table corresponding to each of slave 1 to slave 32 on second FSSB line (second optical connector). Set these parameters only when a servo axis control card with two optical connectors (FSSB lines) is used.

To specify these parameters, follow the same procedure as for the first FSSB line (parameters Nos. 24000 to 24031). Note, however, that the valid data range varies depending on the separate detector interface unit used.

- When the slave is a separate detector interface unit:
  Set 3005, 3006, 3007, and 3008, respectively, for the first (one connected nearest to the CNC), second, third, and fourth separate detector interface units.

24096	Connector number for the first separate detector interface unit

24097	Connector number for the second separate detector interface unit

24098	Connector number for the third separate detector interface unit
24099	Connector number for the fourth separate detector interface unit
24100	Connector number for the fifth separate detector interface unit
24101	Connector number for the sixth separate detector interface unit
24102	Connector number for the seventh separate detector interface unit
24103	Connector number for the eighth separate detector interface unit

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 8

Set a connector number for the connector to which a separate detector interface unit is attached if the separate detector interface unit is to be used. The following table lists the necessary settings. Be sure to specify 0 for connectors not in use.

Correspondence between connectors and connector numbers	
Connector	**Connector number**
JF101	1
JF102	2
JF103	3
JF104	4
JF105	5
JF106	6
JF107	7
JF108	8

(Setting example)

Controlled axis	Connector to which each separate detector interface unit is attached				Parameter setting			
	1st connector	2nd connector	5th connector	6th connector	No. 24096	No. 24097	No. 24100	No. 24101
X1	JF101	-	-	-	1	0	0	0
Y1	-	JF102	-	-	0	2	0	0
Z1	-	-	JF102	-	0	0	2	0
X2	-	JF101	-	-	0	1	0	0
Y2	-	-	-	JF101	0	0	0	1
Z2	-	-	-	-	0	0	0	0
A1	-	-	JF101	-	0	0	1	0
B1	-	-	-	JF102	0	0	0	2
C1	-	JF104	-	-	0	4	0	0
A2	JF102	-	-	-	2	0	0	0
B2	-	JF103	-	-	0	3	0	0
C2	-	-	-	JF103	0	0	0	3

NOTE

1. Specify these parameters when separate detector interface units are used.
2. Parameters Nos. 24096 to 24103 are specified automatically when data is entered on the FSSB setting screen if the FSSB setting mode in use is the automatic setting mode (bit 0 (FMD) of parameter No. 1902 = 0). If the manual setting 2 mode (bit 0 (FMD) of parameter No. 1902) = 1), specify the parameters directly.

24104	ATR value corresponding to connector 1 on the first separate detector interface unit

24105	ATR value corresponding to connector 2 on the first separate detector interface unit
to	to
24111	ATR value corresponding to connector 8 on the first separate detector interface unit

24112	ATR value corresponding to connector 1 on the second separate detector interface unit
to	to
24119	ATR value corresponding to connector 8 on the second separate detector interface unit

24120	ATR value corresponding to connector 1 on the third separate detector interface unit
to	to
24127	ATR value corresponding to connector 8 on the third separate detector interface unit

24128	ATR value corresponding to connector 1 on the fourth separate detector interface unit
to	to
24135	ATR value corresponding to connector 8 on the fourth separate detector interface unit

24136	ATR value corresponding to connector 1 on the fifth separate detector interface unit
to	to
24143	ATR value corresponding to connector 8 on the fifth separate detector interface unit

24144	ATR value corresponding to connector 1 on the sixth separate detector interface unit
to	to
24151	ATR value corresponding to connector 8 on the sixth separate detector interface unit

24152	ATR value corresponding to connector 1 on the seventh separate detector interface unit
to	to
24159	ATR value corresponding to connector 8 on the seventh separate detector interface unit

24160	ATR value corresponding to connector 1 on the eighth separate detector interface unit
to	to
24167	ATR value corresponding to connector 8 on the eighth separate detector interface unit

NOTE

When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1000 to 1046

Each of these parameters sets the value (ATR value) of the address translation table corresponding to each connector on a separate detector interface unit.
The first to fourth separate detector interface units are connected to first FSSB line, and the fifth and eighth separate detector interface units are connected to second FSSB line.
Specify each parameter with a value set in parameter No. 1023 (axis connected to a separate detector interface unit connector) plus 1000.

If a connector attached to a separate detector interface unit is not in use, set 1000 for the connector.

> **NOTE**
> 1 Specify these parameters if one separate detector interface unit connector is shared among two or more axes. They need not be specified if one connector is used by one axis.
> 2 Using these parameters requires setting bit 5 (SSC) of parameter No. 14476 to 1.

Special setting for a separate detector 1

If you want to use one connector of a separate detector for multiple axes, set bit 5 (SSC) of parameter No. 14476 to 1 and parameter Nos. 24104 to 24199.

- Example of special setting for a separate detector 1

For two EGB pairs and common master axis
(EGB slave axis 1 : A axis, EGB dummy axis 1 : C1 axis)
(EGB slave axis 2 : B axis, EGB dummy axis 2 : C2 axis)

No.	14476#5 SSC
	1

No.	1023	24096
X	1	3
Y	2	4
Z	9	0
A	3	0
B	5	0
C1	4	1
C2	6	1

No.	24104	24105	24106	24107	24108	24109	24110	24111
	1004	1006	1001	1002	1000	1000	1000	1000

Special setting for a separate detector 2

When one servo axis use two separate position detectors, two separate detector interface units are required. In the case of a servo card, one of them needs to be 1, 3, 5 or 7th interface unit and the other needs to be 2, 4, 6 or 8th interface unit.

- Example of special setting for a separate detector 2

When all axes recieve the feedback data from two separate position detectors

No.	24096 first ninth	24097 second tenth	24098 third eleventh	24099 fourth twelfth	24100 fith	24101 sixth	24102 seventh	24103 eighth
X1	1	1	0	0	0	0	0	0
Y1	2	0	0	1	0	0	0	0
Z1	3	0	0	0	0	1	0	0
A1	4	0	0	0	0	0	0	1
X2	0	2	1	0	0	0	0	0
Y2	0	3	0	0	1	0	0	0
Z2	0	4	0	0	0	0	1	0
A2	0	0	2	2	0	0	0	0
X3	0	0	3	0	0	2	0	0

No.	24096 first ninth	24097 second tenth	24098 third eleventh	24099 fourth twelfth	24100 fith	24101 sixth	24102 seventh	24103 eighth
Y3	0	0	4	0	0	0	0	2
Z3	0	0	0	3	2	0	0	0
A3	0	0	0	4	0	0	2	0

Alarm and message

Number	Message	Description
SV0456	ILLEGAL CURRENT LOOP	An attempt was made to set the current loop that could not be set. The amplifier pulse module in use does not comply with HIGH SPEED HRV. Or, requirements to control are not satisfied in the system.
SV0458	CURRENT LOOP ERROR	The specified current loop differs from the actual current loop.
SV0459	HI HRV SETTING ERROR	For two axes whose servo axis numbers (parameter No. 1023) are consecutively even and odd numbers, HIGH SPEED HRV control is possible for one axis and impossible for the other.
SV0462	SEND CNC DATA FAILED	The correct data could not be received on a slave side because of the FSSB communication error.
SV0463	SEND SLAVE DATA FAILED	The correct data could not be received in the servo software because of the FSSB communication error.
SV0465	READ ID DATA FAILED	A read of the ID information for the amplifier has failed at power-on.
SV0466	MOTOR/AMP. COMBINATION	The maximum current of an amplifier is different to that of a motor. Or, the connection command for an amplifier is incorrect. The parameter setting is incorrect
SV0468	HI HRV SETTING ERROR(AMP)	An attempt was made to set up HIGH SPEED HRV control for use when the controlled axis of an amplifier for which HIGH SPEED HRV control could not be used.
SV1067	FSSB:CONFIGURATION ERROR(SOFT)	An FSSB configuration error occurred (detected by software). The connected amplifier type is incompatible with the FSSB setting value.
SV5134	FSSB:OPEN READY TIME OUT	In the initialization, the FSSB could not be in an open ready sate. The axis card is thought to be defective.
SV5136	FSSB:NUMBER OF AMP. IS INSUFFICIENT	The number of amplifier identified by the FSSB is insufficient than the number of control axes. Or, the setting of the number of axes or the amplifier connection is in error.
SV5137	FSSB:CONFIGURATION ERROR	An FSSB configuration error occurred. The connecting amplifier type is incompatible with the FSSB setting value.
SV5139	FSSB:ERROR	Servo initialization has not completed successfully. It is probable that an optical cable failed or a connection between the amplifier and another module failed.
SV5197	FSSB:OPEN TIME OUT	The initialization of the FSSB was completed, but it could not be opened. Or, the connection between the CNC and the amplifier is inccorrect.
SV5311	FSSB:ILLEGAL CONNECTION	Different current loops (HRV) are set for FSSB lines. Specify the same current loop for the FSSB lines.
SP1220	NO SPINDLE AMP.	Either the cable connected to a serial spindle amplifier is broken, or the serial spindle amplifier is not connected.
SP1996	ILLEGAL SPINDLE PARAMETER SETTING	The spindle was assigned incorrectly. Check to see the following parameter. (No.3716 or 3717)

Reference item

Manual name	Item name
CONNECTION MANUAL (HARDWARE) (B-64693EN)	Connection to the amplifiers
CONNECTION MANUAL (FUNCTION) (this manual)	Spindle serial output

Diagnosis data

3510	FSSB alarm number

[Data type] Word

Information is output for identifying the location (parameter) and cause of an FSSB-related alarm which has been issued. For the displayed detail numbers and corresponding causes and actions, see the table below.

Detail alarm No.	Parameter number	Cause	Action
120 451 452	-	The FSSB internal status did not change to open.	Check the connection between the CNC and each amplifier. Alternatively, the servo card may be faulty.
140 450	24000 to 24095	The ATR value is inconsistent with the connected slave (servo, spindle, or separate detector).	Set the ATR value corresponding to the connected slave.
271	3717 24000 to 24095	The spindle amplifier number corresponding to the ATR value setting is not set.	Make the spindle amplifier number consistent with the ATR value setting.
272	24000 to 24031 24064 to 24095	The fifth to eighth separate detector is set for the first FSSB line (third FSSB line).	Do not set the fifth to eighth separate detectors for the first FSSB line (third FSSB line).
273	24032 to 24063	The first to fourth (ninth to twelfth) separate detector is set for the second FSSB line.	Do not set the first to fourth (ninth to twelfth) separate detectors for the second FSSB line.
276	24000 to 24095	The setting for a separate detector is made more than once.	Make the setting for each separate detector only once in the servo card.
290	24000 to 24095	The maximum number of slaves per FSSB line is exceeded for an FSSB line of servo HRV2 control.	Reduce the number of slaves to 32 (maximum number of slaves per FSSB line of servo HRV2 control) or less.
291	24000 to 24095	The maximum number of slaves per FSSB line is exceeded for an FSSB line of servo HRV3 control.	Reduce the number of slaves to 15 (maximum number of slaves per FSSB line of servo HRV3 control) or less.
310	1023 24104 to 24199	The servo axis number corresponding to the ATR value setting of a separate detector is not set for parameter No. 1023.	Set the value corresponding to the ATR value setting for parameter No. 1023.
313	1023 14476#5 24104 to 24199	The servo axis number corresponding to the ATR value setting of a separate detector is not set for parameter No. 1023.	Set the value corresponding to the ATR value setting for parameter No. 1023.
314	1023 14476#5 24104 to 24199	The ATR value setting of a separate detector is invalid.	Correct the settings of parameters Nos. 24104 to 24199.
383	-	Manual setting 1 cannot be performed when a separate detector is used.	Disconnect the separate detector. Alternatively, perform manual setting or automatic setting.

Detail alarm No.	Parameter number	Cause	Action
453	-	Servo initialization has not completed successfully.	An optical cable may be faulty or the connection between the amplifier and another module may be incorrect.
454	-	Alarm No. 550 to 556 of diagnosis data No. 3511 occurred.	Check diagnosis data No. 3511.
460	24000 to 24095	The ATR value of a spindle or separate detector is set for a slave which is not connected.	Set the ATR value corresponding to the connected slave.
471	24000 to 24095	Although a separate detector is connected, the separate detector setting is not made.	Set the value for the separate detector in the corresponding parameter.
480	24000 to 24095	In ATR value setting, a servo axis number exceeds 80.	Make settings so that any servo axis number does not exceed 80.

3511	FSSB alarm number

[Data type] Word axis

Information is output for identifying the location (parameter) and cause of an FSSB-related alarm which has been issued. For the displayed detail numbers and corresponding causes and actions, see the table below.

Detail alarm No.	Parameter number	Cause	Action
210	24096 to 24103	Although a separate detector is not set, a value is set in parameter No. 24096 to 24103.	Set parameter Nos. 24096 to 24103 to all 0.
220	1023	An unavailable servo axis number is set.	Change the servo axis number.
221	1023	A servo axis number is set more than once.	Change the servo axis number.
250	24096 to 24103	For a specific servo axis, two or more separate detectors are used and the paired separate detectors are two of the first, third, fifth, and seventh units or the second, fourth, sixth, and eighth units.	To use two separate detectors for a specific servo axis, one separate detector must have an odd number and the other must have an even number. Three or more separate detectors cannot be used.
270	1023 24000 to 24095	・ The servo axis number corresponding to the ATR value setting is not set for parameter No. 1023. ・ An unavailable servo axis number is set. ・ A servo axis number is set more than once.	Check the conditions on the left.
292	1023 2013#0	For an FSSB line of servo HRV3 control, only the following servo axis numbers can be used: (1 + 8n, 2 + 8n, 3 + 8n, 4 + 8n (n = 0, 1, ···, 9))	For the FSSB line of servo HRV3 control, set the servo axis numbers on the left.
311	24096 to 24103	A connector number is invalid.	Specify a value between 0 and 8.
314	24096 to 24103	A connector number is set more than once.	Make setting so that each connector number is used only once for one separate detector.
350	2013#0 2014#0	Different current loops (HRV) are used for FSSB lines.	Set the same current loop (HRV) for the FSSB lines.

Detail alarm No.	Parameter number	Cause	Action
360	1023 2013#0 2014#0	Different current loops (HRV) are set for the first and second FSSB lines and parameter No. 1023 setting is invalid.	Set servo axis numbers so that each set of (1 to 6), (9 to 14), (17 to 22), (25 to 30), (33 to 38), and (41 to 46) is set for the same FSSB line.
370	1902#0 1902#1 2013#0 2014#0	When servo HRV3 control is set, manual setting 1 cannot be performed.	To set servo HRV3 control, perform manual setting or automatic setting.
380	1023	When a servo axis number is skipped, manual setting 1 cannot be performed.	Set servo axis numbers without skipping any number.
382	1023	An attempt was made to perform manual setting 1 though the maximum number of controlled axes per FSSB line is exceeded.	Reduce the number of connected servo axes to the maximum number of controlled axes or less.
470	24000 to 24095	An ATR value is set more than once.	Set each ATR value only once.
481	1023 24000 to 24095	A servo axis number is inconsistent with the ATR value setting or the servo motor having a servo axis number is not connected.	Check whether the value set in parameter No. 1023 is consistent with ATR value setting and whether the servo motor corresponding to each servo axis number is connected.
520	2165	At power-on, amplifier ID information could not be read.	Check the connection between the CNC and each amplifier. Alternatively, an amplifier may be faulty.
550	1023 24000 to 24095	The ATR value setting is inconsistent with the servo axis number setting.	Make the value set in parameter No. 1023 consistent with the ATR value setting.
551	24000 to 24095	The number of ATR value settings exceeds the number of slaves connected to the CNC.	Make as many settings as the number of slaves connected to the CNC.
552	1023	An unavailable servo axis number is set.	Change the servo axis number.
553	1023	A servo axis number is set more than once.	Change the servo axis number.
554	24096 to 24103	A value is set in parameters Nos. 24096 to 24103 though no separate detector is connected.	Set parameters Nos. 24096 to 24103 to all 0.
555 557 558	2165	The maximum current of an amplifier (parameter No. 2165) differs from that of a motor.	Set the maximum current of the amplifier (parameter No. 2165) to that of the motor.
1023	1023	An invalid servo axis number is set.	Set a correct servo axis number.

3513	FSSB alarm number (spindle axis)

[Data type] Word spindle

Information is output for identifying the location (parameter) and cause of an FSSB-related alarm which has been issued.

For the displayed detail numbers and corresponding causes and actions, see the table below.

Detail alarm No.	Parameter number	Cause	Action
271	3717 24000 to 24095	An ATR value is set more than once.	Make each spindle amplifier consistent with the ATR value setting.

Detail alarm No.	Parameter number	Cause	Action
381	3717	When a spindle amplifier number is skipped, manual setting 1 cannot be performed.	Set spindle amplifier numbers without skipping any number.

1.4.4 Temporary Absolute Coordinate Setting

Overview

In the full closed system with an inner absolute position Pulsecoder (serial Pulsecoder) and an incremental scale, the position is set by using absolute position data from the inner absolute position Pulsecoder at the power on sequence. After that, the position is controlled with incremental data from the incremental scale. The position just after power on sequence is rough, and the manual reference position return is required to get the accurate position.
With this function, the position at the power on is rough, but the following functions are available before the reference position return.
- Stroke limit check
- Position switch

Please be careful this function does not make an incremental scale work as an absolute position detector.

Also in using the linear-scale with distance-coded reference marks (serial), these function are not available.

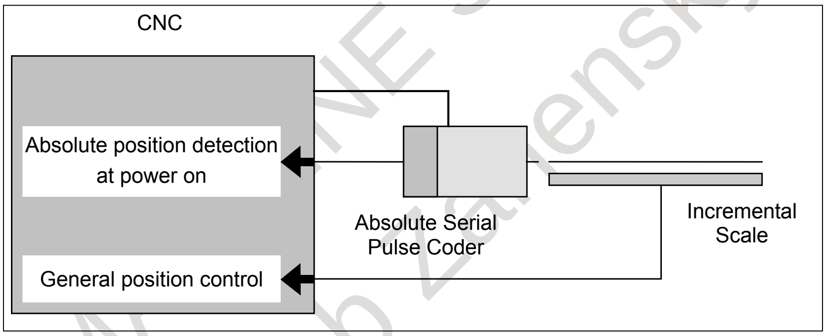


Fig. 1.4.4 (a) The system with the Temporary Absolute Coordinate Setting

In using the temporary absolute coordinate, the coordinate is same as the reference position return has not been performed.
The reference position return is required to get the accurate position after the power on sequence.

The reference position establishment signals ZRF1 to ZRF8<Fn120> are not set to 1 until the manual reference point return is performed.
The diagnosis data No.304 (reference counter value) are also not displayed until the manual reference point return is performed.

- Parameter setting

1 Please set the following parameters to the axis that uses this function.
  - The bit 1 (OPTx) of parameter No. 1815 = 1.
  - The bit 5 (APCx) of parameter No. 1815 = 1.
  - The bit 7 (XIAx) of parameter No. 2011 = 1.

2 Please set the flexible feed gear for the built-in position detector (parameters Nos. 1874 and 1875).

- Setting of the temporary absolute coordinate

Please set the temporary absolute coordinate according to the following procedures.

1 Perform the reference position return and move the reference position.

2 Turn off the power then turn on the power.
3 In MDI mode, bit 4 (APZx) of parameter No. 1815 is set to 1.

> **NOTE**
> In the above operations from 1 to 3, please do not move the axis from the reference position. If the axis is moved from the reference position, the temporary absolute coordinate is not correctly set.

- On the Difference of specifications with FS16*i*/FS18*i*/FS21*i*

With the FS16*i*/FS18*i*/FS21*i*, when bit 1 (XZF) of parameter No. 1807 is set to 0, the coordinate system is established.
At this time, the reference position establishment signals <Fn120> are set to 1, which indicates that the reference position has been established.
Diagnosis data No. 304 (reference counter value) is also displayed.

With the FS0*i*, the built-in absolute Pulsecoder is used to roughly determine the coordinate position without establishing the coordinate system.
At this time, the reference position establishment signals <Fn120> are set to 0, which indicates that the reference position has not been established.
Diagnosis data No. 304 (reference counter value) is not also displayed.

When manual reference position return by the scale of the full closed system is completed, the accurate coordinate system is established.
At this time, the reference position establishment signals <Fn120> are set to 1, which indicates that the reference position has been established.
Diagnosis data No. 304 (reference counter value) is also displayed.

With the FS0*i*, bit 1 (XZF) of parameter No. 1807 is not provided.

- Axis synchronous control

In case of the synchronization establishment of the axis synchronous control, when the position is set by this function, the synchronization establishment is executed.

Signal

Reference position establishment signal ZRF1 to ZRF8<Fn120>

[Classification] Output signal

[Function] Notify the system that the reference position has been established.
A reference position establishment signal is provided for each axis. The number appended to each signal indicates the number of the controlled axis.
ZRFx
x : 1 : 1st-axis reference position establishment signal
2 : 2nd-axis reference position establishment signal
3 : 3rd-axis reference position establishment signal
:
:

[Output cond.] The signals are set to 1 in the following case:
- When the reference position is established after manual reference position return
- When the reference position is established using the absolute-position detector at initial power-on

When the reference position is lost, the signals are set to “0”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn120	ZRF8	ZRF7	ZRF6	ZRF5	ZRF4	ZRF3	ZRF2	ZRF1

Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx	APZx			OPTx	

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#1 OPTx The separate position detector is:
0: Not to be used (semi-closed system)
1: To be used (full-closed system)

> **NOTE**
> 1 In case of using the absolute position detector (bit 5 (APCx) of parameter No.1815 is set to 1), please set the following parameters that correspond to the absolute position detector which is actually used.
> - parameter No.1815#6, No.1815#0, No.1817#3, No.1868, No.2275#1, No.2394
>
> If these parameters are not set correctly, the machine coordinates are not correctly established at power-on.
> 2 In case of using this function, this parameter is set to 1.

#4 APZx Machine position and position on absolute position detector when the absolute position detector is used
0 : Not corresponding
1 : Corresponding

> **NOTE**
> 1 If the following parameters are modified, the bit 4 (APZ) of parameter No. 1815 will be changed to 0.
> No. 1803#7, No. 1815#1, No. 1820, No. 1821, No. 1822, No. 1823, No. 1874, No. 1875, No. 2022, No. 2084, No. 2085
> 2 The bit 4 (APZ) of parameter No. 1815 is kept value 1 even if the grid shift value is modified.

#5 APCx Position detector is
0 : Other than an absolute position detector.
1 : An absolute position detector (absolute Pulsecoder).

NOTE
1 In case of using this function, this parameter is set to 1.
2 When the servo axis is disabled (bit 4 (KSVx) of parameter No. 11802 is set to 1), an absolute position detector cannot be used (bit 5 (APCx) of parameter No. 1815 cannot be set to 1). If an absolute position detector is used, alarm SV0301, "APC ALARM; COMMUNICATION ERROR" is issued.

1874	Numerator of the flexible feed gear for the built-in position detector
1875	Denominator of the flexible feed gear for the built-in position detector

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 1 to 32767

When using temporary absolute coordinate setting, set the flexible feed gear for the built-in position detector on each axis. The settings are as follows:

$$\frac{\text{No.1874}}{\text{No.1875}} = \frac{\text{Number of position feedback pulses per motor revolution}}{1{,}000{,}000}$$

NOTE
When parameter No. 1874 or 1875 is set to 0, both parameters are assumed to be 1.

	#7	#6	#5	#4	#3	#2	#1	#0
2011	XIAx							

[Input type] Parameter input
[Data type] Bit axis

#7 **XIAx** Temporary absolute coordinate setting is:
0: Not used.
1: Used.

NOTE
1 When temporary absolute coordinate setting is used, bit 1 (OPTx) of parameter No. 1815, bit 5 (APCx) of parameter No. 1815, parameter No. 1874, and parameter No. 1875 must be set.
2 The setting of this parameter becomes effective after the power is turned off then back on.

1.4.5 Diagnosis data display of detection unit

Explanation

Diagnosis data No.3740 displays detection unit of each axis. Diagnosis data No.3741 displays machine unit of each axis.

(Example 1)
If detection unit of X axis which is linear axis (metric system machine) is 0.0001[mm], diagnosis data No.3740 displays 0.0001.
Bit 0 (MM) of diagnosis data No.3741 is set to 1 because machine unit of X axis is [mm] (Fig. 1.4.5 (a)).
(Example 2)
If detection unit of C axis which is rotary axis is 0.001[degree], diagnosis data No.3740 displays 0.001.
Bit 2 (DEG) of diagnosis data No.3741 is set to 1 because machine unit of C axis is [degree] (Fig. 1.4.5 (a)).

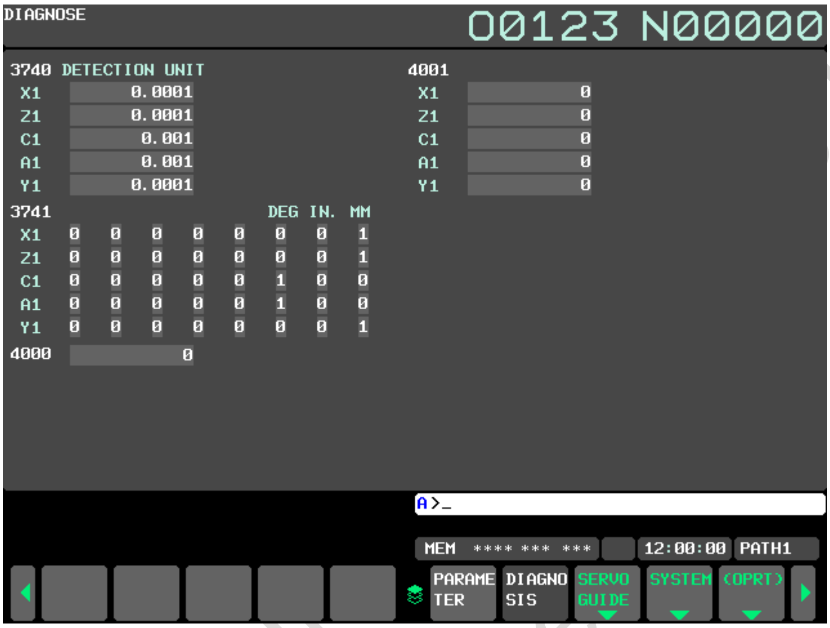


Fig. 1.4.5 (a) Diagnosis screen (metric system machine) (10.4” display unit)

(Example 3)
If detection unit of X axis which is linear axis (inch system machine) is 0.00001[inch], diagnosis data No.3740 displays 0.00001.
Bit 1 (IN.) of diagnosis data No.3741 is set to 1 because machine unit of X axis is [inch] (Fig. 1.4.5 (b)).

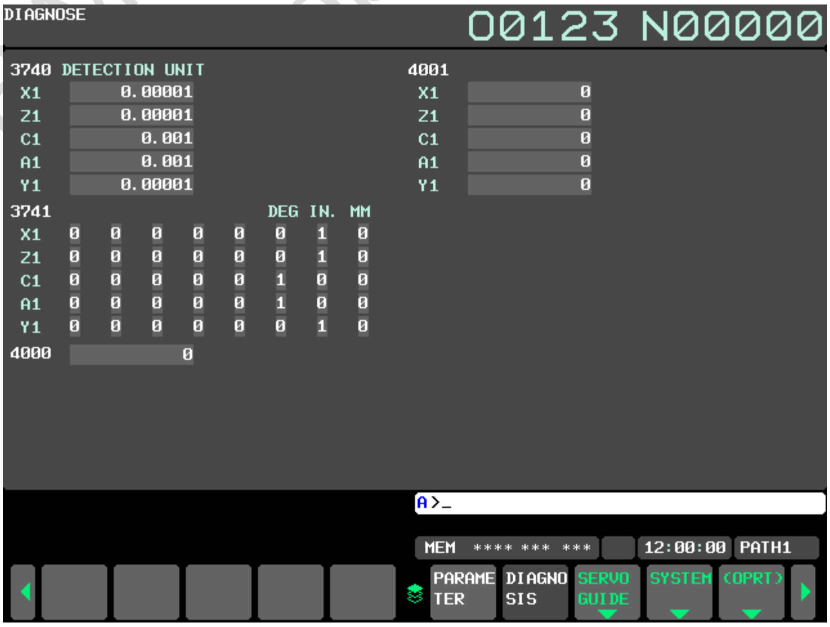


Fig. 1.4.5 (b) Diagnosis screen (inch system machine) (10.4” display unit)

- Servo axis

Diagnosis data No.3740 displays detection unit which is calculated by the following equation.

Detection unit = Least command increment / CMR

- Cs contour control axis

Detection unit of Cs contour control axis is set by bit 0 (CS360M) of parameter No.4005. Diagnosis data No.3740 displays the following value. Diagnosis data No.3740 is displayed 0.0 by the hypothetical Cs axis (bit 7 (CDMx) of parameter No.3740 = 1).

Table 1.4.5 (a) Detection unit of Cs contour control axis

Bit 0 (CS360M) of parameter No.4005	Diagnosis data No.3740
0	0.001
1	0.0001

- Spindle positioning axis

Detection unit of spindle positioning axis is set by gear ratio between spindle and position coder (parameter Nos.3721, 3722). Diagnosis data No.3740 displays the following value.

Table 1.4.5 (b) Detection unit of spindle positioning axis

Gear ratio of spindle to position coder	Diagnosis data No.3740
1:1	0.088 (1x360/4096)
1:2	0.176 (2x360/4096)
1:4	0.352 (4x360/4096)
1:8	0.703 (8x360/4096)
…	…
1:N	(Nx360/4096)

Diagnosis data

3740	Detection unit

[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Valid data range] 0.0 to 99.99
This diagnosis data displays detection unit.

	#7	#6	#5	#4	#3	#2	#1	#0
3741						DEG	IN.	MM

[Data type] Bit axis

#0 MM Detection unit is:
0: Not mm.
1: mm.

#1 IN. Detection unit is:
0: Not inch.
1: inch.

#2 DEG Detection unit is:
0: Not degree.
1: degree.

1.5 SETTINGS RELATED WITH COORDINATE SYSTEMS

1.5.1 Machine Coordinate System

Overview

The point that is specific to a machine and serves as the reference of the machine is referred to as the machine zero point. A machine tool builder sets a machine zero point for each machine.
A coordinate system with a machine zero point set as its origin is referred to as a machine coordinate system.
A machine coordinate system is set by performing manual reference position return after power-on. A machine coordinate system, once set, remains unchanged until the power is turned off.

Format

M

(G90)G53 IP _ P1;
IP_ : Absolute command dimension word
P1 : Enables the high-speed G53 function.

(G90)G53.2 G01 IP__F__;
IP_ : Absolute command dimension word
F_ : Feedrate

T

G53 IP _ P1;
IP_ : Absolute command dimension word
P1 : Enables the high-speed G53 function.

G53.2 G01 IP__F__;
IP_ : Absolute command dimension word
F_ : Feedrate

Explanation

- Selecting a machine coordinate system (G53)

When a command is specified the position on a machine coordinate system, the tool moves to the position by rapid traverse. G53, which is used to select a machine coordinate system, is a one-shot G code; that is, it is valid only in the block in which it is specified on a machine coordinate system. Specify an absolute command for G53. When an incremental command is specified, the G53 command is ignored. When the tool is to be moved to a machine-specific position such as a tool change position, program the movement in a machine coordinate system based on G53.

- High-speed G53 function

The high-speed G53 function enables the rapid traverse block overlap function between the machine coordinate system selection command (G53) block and positioning (rapid traverse) command (G00) block, which allows the next rapid traverse command (G00) to be executed without decelerating and stopping the tool at the end of the machine coordinate system selection command (G53). This function enables high-speed positioning even when the machine coordinate system selection command (G53) is used.
The high-speed G53 function is enabled by specifying P1 in the G53 block.

- Selecting a machine coordinate system with feedrate (G53.2)

Positioning of machine coordinate system at a feed rate is available with the command of G53.2. The feed rate can be used in the modal of G01.

> **NOTE**
> The selecting a machine coordinate system with feedrate (G53.2) is an option. To use this function, an option of the selecting a machine coordinate system with feedrate is required.

G53.2 is one shot G code. Moreover, the tool offset value is temporarily canceled by G53.2 command like G53.
As for the time constant, a usual feed rate are used. At the point of G53.2 command, the axis decelerates to a stop once like G53 command.
Feed per minute, feed per revolution, and inverse time feed of G codes of 05 group are available.
When group 01 G-code except G00/G01 is commanded with G53.2, the alarm PS5372, "IMPROPER MODAL G-CODE" is generated. And, when G53.2 is commanded while group 01 Modal G-code is the one except G00/G01, the alarm PS5372 is generated.

Limitation

- Cancel of the compensation function

When the G53 command is specified, cancel the compensation functions such as the cutter compensation, tool length offset, tool-nose radius compensation, and tool offset.

- G53 specification immediately after power-on

Since the machine coordinate system must be set before the G53 command is specified, at least one manual reference position return or automatic reference position return by the G28 command must be performed after the power is turned on. This is not necessary when an absolute-position detector is attached.

Specification in the same block

M

Commands G50/G51, G50.1/G51.1, and G68/G69 cannot be specified in the same block where the G53 command is specified.

T

Commands G50/G51 (except for G code system A), G50.1/G51.1, and G68.1/G69.1 cannot be specified in the same block where the G53 command is specified.

Reference

- Setting a machine coordinate system

When manual reference position return is performed after power-on, a machine coordinate system is set so that the reference position is at the coordinate values of (α, β) set using parameter No.1240.

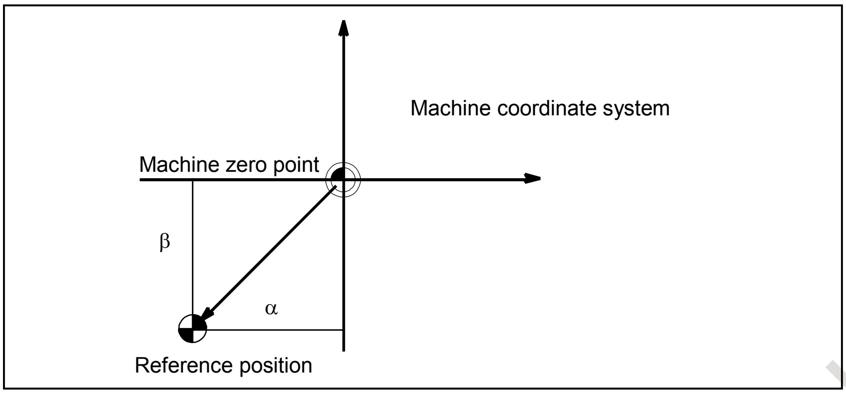


Fig. 1.5.1 (a)

Example (selecting a machine coordinate system with feedrate)

N1 G90 G01 ;
N2 G53.2 X50.0 Y100.0 F1000 ; Absolute command with feedrate F1000
N3 G53.2 X150.0 F500 ; Absolute command with feedrate F500

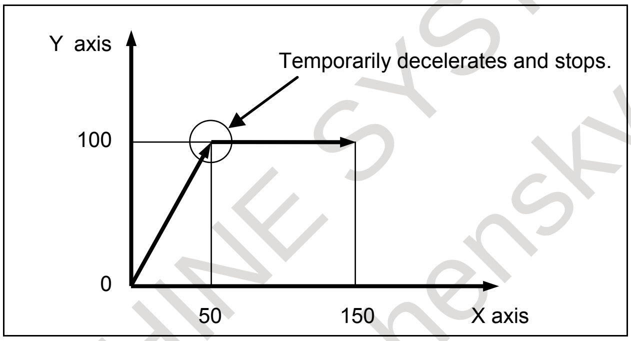


Fig. 1.5.1 (b)

Parameter

1240	Coordinate value of the reference position in the machine coordinate system

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the reference position in the machine coordinate system.

Alarm and message

Number	Message	Description
PS5372	IMPROPER MODAL G-CODE (G53.2)	In the G53.2 block, a G code in group 01 other than G00 or G01 is specified. Alternatively, G53.2 is specified when the modal G code in group 01 is other than G00 or G01.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Machine coordinate system

1.5.2 Workpiece Coordinate System/Addition of Workpiece Coordinate System Pair

Overview

A coordinate system used for machining a workpiece is referred to as a workpiece coordinate system. A workpiece coordinate system is to be set with the CNC beforehand (setting a workpiece coordinate system).
A machining program sets a workpiece coordinate system (selecting a workpiece coordinate system).
A set workpiece coordinate system can be changed by shifting its origin (changing a workpiece coordinate system).

1.5.2.1 Workpiece coordinate system

> **NOTE**
> To use the workpiece coordinate system, set 0 in bit 0 (NWZ) of parameter No.8136.

- Setting a workpiece coordinate system

A workpiece coordinate system can be set using one of three methods:

(1) Method using a workpiece coordinate system setting G code
A workpiece coordinate system is set by specifying a value in the program after a workpiece coordinate system setting G code.

(2) Automatic setting
If bit 0 (ZPR) of parameter No. 1201 is set to 1, a workpiece coordinate system is automatically set when manual reference position return is performed.
This function is, however, disabled when the workpiece coordinate system is being used.

(3) Method using a workpiece coordinate system selection G code
Six workpiece coordinate systems can be set beforehand using the MDI unit. Program commands G54 to G59 can be used to select the work axis to be used.

When using an absolute command, establish the workpiece coordinate system in any of the above ways.

> ⚠ **CAUTION**
> The established workpiece coordinate system depends on diameter programming or radius programming.

- Selecting a workpiece coordinate system

The user can choose from set workpiece coordinate systems as described below.

(1) Selecting a workpiece coordinate system set by workpiece coordinate system setting G code (G92, G50) or automatic workpiece coordinate system setting
Once a workpiece coordinate system is selected, absolute commands work with the workpiece coordinate system.

(2) Choosing from six workpiece coordinate systems set using the MDI unit
By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.

G54.....Workpiece coordinate system 1   G55..... Workpiece coordinate system 2
G56.....Workpiece coordinate system 3   G57..... Workpiece coordinate system 4
G58.....Workpiece coordinate system 5   G59..... Workpiece coordinate system 6

Workpiece coordinate systems 1 to 6 are established after reference position return after the power is turned on. When the power is turned on, G54 coordinate system is selected as default.

When bit 2 (G92) of parameter No. 1202 is set to 1, executing the G code command for coordinate system setting (G92 <M series>, G50 <Tseries> (G92 with G code system B/C for T series)) results in the issue of alarm PS0010, “IMPROPER G-CODE”. This is designed to prevent the user from confusing coordinate systems.

> ⚠ **CAUTION**
> The set workpiece zero point offset value depends on diameter programming or radius programming.

- Changing workpiece coordinate system

The six workpiece coordinate systems specified with G54 to G59 can be changed by changing an external workpiece zero point offset value or workpiece zero point offset value.
Three methods are available to change an external workpiece zero point offset value or workpiece zero point offset value.

(1) Inputting from the MDI unit
(2) Programming (using a programmable data input G code or a workpiece coordinate system setting G code)
(3) Using the external data input function
An external workpiece zero point offset value can be changed by input signal to CNC. Refer to machine tool builder's manual for details.

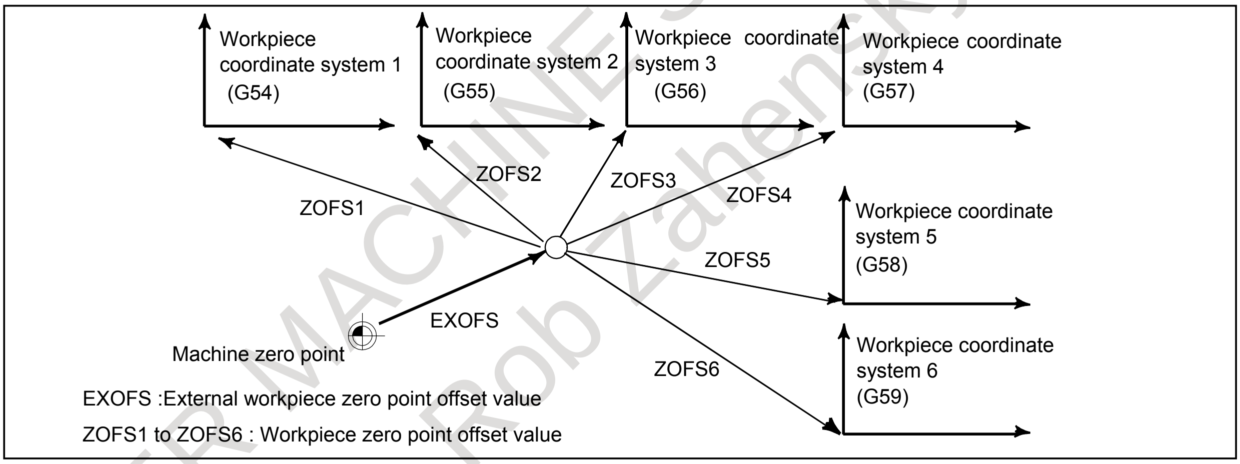


Fig. 1.5.2.1 (a) Changing an external workpiece zero point offset value or workpiece zero point offset value

Format

- Changing by inputting programmable data

G10 L2 Pp IP_;
p=0 : External workpiece zero point offset value
p=1 to 6: Workpiece zero point offset value correspond to workpiece coordinate system 1 to 6
IP_: For an absolute command, workpiece zero point offset for each axis.
For an incremental command, value to be added to the set workpiece zero point offset for each axis (the result of addition becomes the new workpiece zero point offset).

- Changing by setting a workpiece coordinate system

M

G92 IP_;

T

G50 IP_;

Explanation

- Changing by inputting programmable data

By specifying a programmable data input G code, the workpiece zero point offset value can be changed for each workpiece coordinate system.

- Changing by setting a workpiece coordinate system

By specifying a workpiece coordinate system setting G code, the workpiece coordinate system (selected with a code from G54 to G59) is shifted to set a new workpiece coordinate system so that the current tool position matches the specified coordinates (IP_).
Then, the amount of coordinate system shift is added to all the workpiece zero point offset values. This means that all the workpiece coordinate systems are shifted by the same amount.

> ⚠ **CAUTION**
> When a coordinate system is set with G code command for workpiece coordinate system setting after an external workpiece zero point offset value is set, the coordinate system is not affected by the external workpiece zero point offset value. When G92X100.0Z80.0; is specified, for example, the coordinate system having its current tool reference position at X = 100.0 and Z = 80.0 is set.

T

If IP is an incremental command value, the work coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position. (Coordinate system shift)
When bit 0 (WAB) of parameter No. 11279 is set to 1, the setting of the workpiece coordinate system is executed with absolute value even if the work coordinate system setting (G92) is commanded during the incremental mode (G91) in the G-code system B/C of the lathe system.

Notes

- Manual reference position return

When the manual reference position return is performed, the value of the workpiece coordinate system set by G code (G92, or G50 for the lathe system G-code system A) is cleared to 0. And when the automatic reference position return (G28) is performed, this workpiece coordinate system is not cleared to 0 but stored.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11279								WAB

[Input type] Parameter input
[Data type] Bit

#0 WAB When the incremental mode (G91) is selected in the G-code system B/C of the lathe system, if the setting of the workpiece coordinate system (G92) is commanded, the setting of the workpiece coordinate system is executed with:

0: Incremental value.
1: Absolute value.

1.5.2.2 Workpiece coordinate system preset

NOTE

To use the workpiece coordinate system preset function, set 0 in bit 1 (NWC) of parameter No. 8136.

Explanation

In reset state, execute Manual reference position return, a workpiece coordinate system is shifted by the workpiece zero point offset value from the machine coordinate system zero point. Suppose that the manual reference position return operation is performed when a workpiece coordinate system is selected with G54. In this case, a workpiece coordinate system is automatically set which has its zero point displaced from the machine zero point by the G54 workpiece zero point offset value; the distance from the zero point of the workpiece coordinate system to the reference position represents the current position in the workpiece coordinate system.

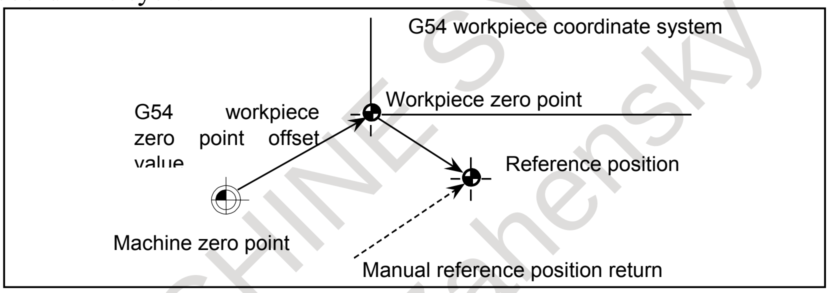


Fig. 1.5.2.2 (a)

If an absolute position detector is provided, the workpiece coordinate system automatically set at power-up has its zero point displaced from the machine zero point by the G54 workpiece zero point offset value. The machine position at the time of power-up is read from the absolute position detector and the current position in the workpiece coordinate system is set by subtracting the G54 workpiece zero point offset value from this machine position. The workpiece coordinate system set by these operations is shifted from the machine coordinate system using the commands and operations listed below.

(a) Manual intervention performed when the manual absolute signal is off
(b) Move command executed in the machine lock state
(c) Movement by handle interrupt
(d) Operation using the mirror image function
(e) Shifting the workpiece coordinate system by setting the local coordinate system or workpiece coordinate system

In the case of (a) above, the workpiece coordinate system is shifted by the amount of movement during manual intervention.

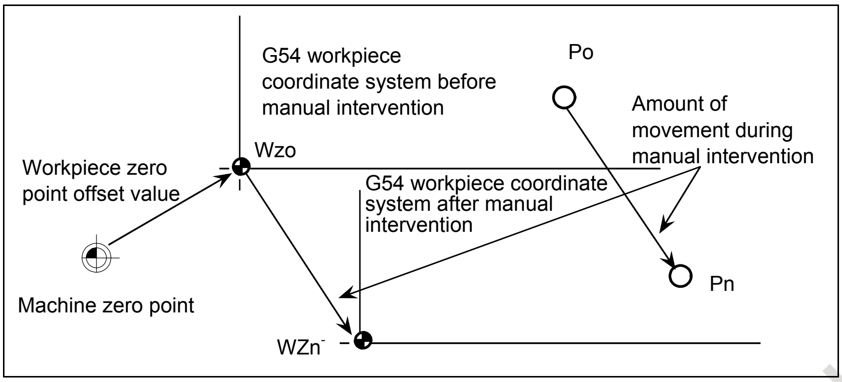


Fig.1.5.2.2 (b)

In the operation above, like execute Manual reference position return, a workpiece coordinate system once shifted can be preset using G code specification (G92.1/G50.3 (G code system A (T series))) or MDI operation (Refer to OPERATOR'S MANUAL (Common to Lathe System / Machining Center System) (B-64694EN) "Workpiece Coordinate System Preset" in "SETTING AND DISPLAYING DATA") to a workpiece coordinate system displaced by a workpiece zero point offset value from the machine zero point.
Bit 3 (PPD) of parameter No. 3104 specifies whether to preset relative coordinates as well as absolute coordinates.
When no workpiece coordinate system option (G54 to G59) is not provided (1 is set in bit 0 (NWZ) of parameter No. 8136), the workpiece coordinate system is preset to the coordinate system with its zero point placed at the reference position.

Limitation

M

- Cutter compensation, tool length compensation, tool offset

When using the workpiece coordinate system preset function, cancel compensation modes: cutter compensation, tool length compensation, and tool offset. If the function is executed without cancelling these modes, compensation vectors are cancelled.

T

- Tool-nose radius compensation, tool offset

When using the workpiece coordinate system preset function, cancel compensation modes: tool-nose radius compensation and tool offset. If the function is executed without cancelling these modes, compensation vectors are cancelled.

- Program restart

The workpiece coordinate system preset function is not executed during program restart.

- Prohibited modes

Do not use the workpiece coordinate system preset function when the scaling, coordinate system rotation, programmable mirror image, or drawing copy mode is set.

1.5.2.3 Adding workpiece coordinate systems (G54.1 or G54)

M

Besides the six workpiece coordinate systems (standard workpiece coordinate systems) selectable with G54 to G59, 48 or 300 additional workpiece coordinate systems (additional workpiece coordinate systems) can be used.

> **NOTE**
> 1 To use the additional workpiece coordinate systems (48 pairs), set 0 in bit 0 (NWZ) of parameter No. 8136 and 0 in bit 2 (NWN) of parameter No. 8136.
> 2 The additional workpiece coordinate systems (300 pairs) is an option. To use this function, an option of the additional workpiece coordinate systems (300 pairs) is required.

Explanation

- Selecting the additional workpiece coordinate systems

When a P code is specified together with G54.1 (G54), the corresponding coordinate system is selected from the additional workpiece coordinate systems (1 to 48 or 1 to 300).
A workpiece coordinate system, once selected, is valid until another workpiece coordinate system is selected. Standard workpiece coordinate system 1 (selectable with G54) is selected at power-on.

G54.1 P1.........Additional workpiece coordinate system 1
G54.1 P2.........Additional workpiece coordinate system 2
:
G54.1 P48.......Additional workpiece coordinate system 48
:
G54.1 P300.....Additional workpiece coordinate system 300

As with the standard workpiece coordinate systems, the following operations can be performed for a workpiece zero point offset in an additional workpiece coordinate system:

(1) The workpiece zero point offset value setting screen can be used to display and set a workpiece zero point offset value.
(2) The G10 function enables a workpiece zero point offset value to be set by programming.
(3) A custom macro allows a workpiece zero point offset value to be handled as a system variable.
(4) Workpiece zero point offset data can be entered or output as external data.
(5) The PMC window function enables workpiece zero point offset data to be read as program command modal data.

- Setting the workpiece zero point offset value in the additional coordinate systems (G10)

When a workpiece zero point offset value is specified using an absolute value, the specified value is the new offset value. When it is specified using an incremental value, the specified value is added to the current offset value to obtain a new offset value.

Limitation

- Specifying P codes

A P code must be specified after G54.1 (G54). If G54.1 is not followed by a P code in the same block, additional workpiece coordinate system 1 (G54.1P1) is assumed.
If a value not within the specifiable range is specified in a P code, an alarm PS0030, “ILLEGAL OFFSET NUMBER” is issued.
P codes other than workpiece offset numbers cannot be specified in a G54.1 (G54) block.

Example 1) G54.1G04P1000;
Example 2) G54.1M98P48;

1.5.2.4 Automatic coordinate system setting

When bit 0 (ZPR) of parameter No. 1201 for automatic coordinate system setting is 1, a coordinate system is automatically determined when manual reference position return is performed.
Once α, β, and γ are set with parameter No. 1250, a workpiece coordinate system is set upon reference position return so that the base point on the tool holder or the tip of the basic tool is positioned at X = α, Y = β, and Z = γ.
This processing occurs as if the following are specified at the reference position:
Note that this cannot be used when the workpiece coordinate system is provided (when 0 is set in bit 0 (NWZ) of parameter No. 8136).

M

G92 X<u>α</u> Y<u>β</u> Z<u>γ</u> ;

T

G50 X<u>α</u> Z<u>γ</u> ;

When the setting of a workpiece coordinate system shift amount is other than 0, a workpiece coordinate system shifted by the amount is set.

1.5.2.5 Workpiece coordinate system shift

T

Explanation

When the coordinate system actually set by the G50 command or the automatic system setting deviates from the programmed workpiece coordinate system, the set coordinate system can be shifted. Set the desired shift amount in the work coordinate system shift memory.

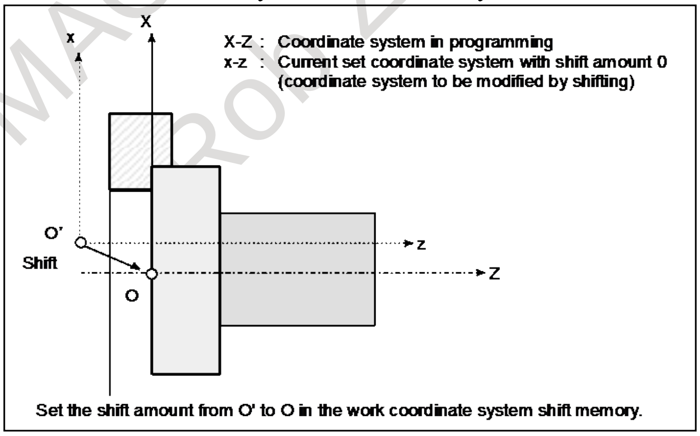


Fig. 1.5.2.5 (a) Workpiece coordinate system shift

Format

- Changing the workpiece coordinate system shift amount

G10 P0 IP_;
IP : Settings of an axis address and a workpiece coordinate system shift amount

⚠ CAUTION

1. A single block can contain a combination of X, Y, Z, C, U, V, W, and H (in G-code system A). In this case, if commands are specified for the same axis, whichever appears later becomes valid.
2. Be sure not to specify other G codes for the same block. When specified, only modal information is updated.
3. Be sure not to specify any auxiliary functions / spindle speed functions / tool functions or 2nd auxiliary functions for the same block. When specified, alarm PS1144 "G10 FORMAT ERROR" is issued.
4. When it commanded with bit 6 (NWS) of parameter No. 1201 is set to 1, alarm PS0010, "IMPROPER G-CODE" is issued.

Limitation

- Shift amount and coordinate system setting command

Specifying a coordinate system setting command (G50 or G92) invalidates the shift amount that has already been set.

Example)

When G50X100.0Z80.0; is specified, a coordinate system is set so that the current base position of the tool is at X =100.0 and Z = 80.0, regardless of which value has been set for the workpiece coordinate system shift amount.

- Shift amount and coordinate system setting

After a shift amount is set, when automatic coordinate system setting is performed upon manual reference position return, the set coordinate system is immediately shifted by the set amount.

- Diameter and radius values

The workpiece coordinate system shift amount depends on diameter programming or radius programming.

Example)

Although the base point should be positioned at X = ϕ120.0 (diameter value) and Z = 70.0 from the workpiece zero point, the actual position is at X = ϕ121.0 and Z = 69.0 from the zero point. Set a shift amount as shown below:

X=1.0, Z=-1.0

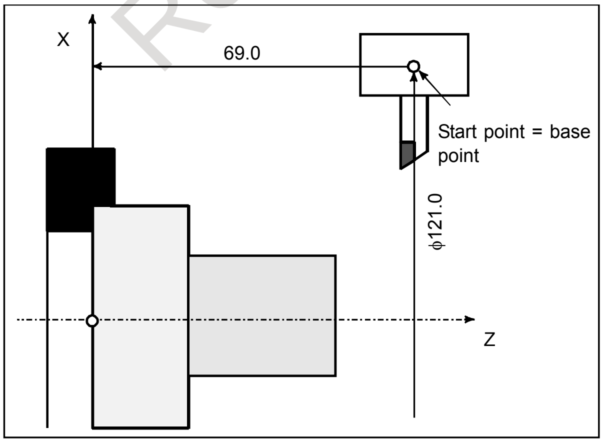


Fig. 1.5.2.5 (b)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1201	WZR	NWS						ZPR
	WZR							ZPR

[Input type] Parameter input
[Data type] Bit path

#0 ZPR Automatic setting of a coordinate system when the manual reference position return is performed
0: Not set automatically
1: Set automatically

> **NOTE**
> ZPR is valid while a workpiece coordinate system function is not provided (bit 0 (NWZ) of parameter No.8136 is 1). If a workpiece coordinate system function is provided, making a manual reference position return always causes the workpiece coordinate system to be established on the basis of the workpiece zero point offset (parameters Nos. 1220 to 1226), irrespective of this parameter setting.

#6 NWS The workpiece coordinate system shift amount setting screen is:
0: Displayed
1: Not displayed

> **NOTE**
> When workpiece coordinate system shift amount setting screen is not displayed, a workpiece coordinate system shift amount modification using G10P0 cannot be made. If G10P0 is commanded, alarm PS0010, "IMPROPER G-CODE" is issued.

#7 WZR If the CNC is reset by the reset key on the MDI unit, external reset signal, reset and rewind signal, or emergency stop signal when bit 6 (CLR) of parameter No. 3402 is set to 0, the G code of group number 14 (workpiece coordinate system) is:
0: Placed in the reset state (not returned to G54).
1: Placed in the cleared state (returned to G54).

> **NOTE**
> 1 When the 3-dimensional conversion mode is set, and bit 2 (D3R) of parameter No. 5400 is set to 1, the G code is placed in the reset state, regardless of the setting of this parameter.
> 2 When bit 6 (CLR) of parameter No. 3402 is set to 1, whether to place the G code in the reset state depends on bit 6 (C14) of parameter No. 3407.

	#7	#6	#5	#4	#3	#2	#1	#0
1202						G92	EWS	EWD
						G92		EWD

[Input type] Parameter input
[Data type] Bit path

#0 EWD The shift direction of the workpiece coordinate system is:
0: The direction specified by the external workpiece zero point offset value
1: In the opposite direction to that specified by the external workpiece zero point offset value

#1 EWS The external workpiece zero point offset is made:
0: Valid
1: Invalid

NOTE
When the external workpiece zero point offset is made invalid, the following operation results:
1 As the external workpiece zero point offset on the workpiece zero point offset setting screen, a workpiece coordinate system shift amount is displayed.
2 Data keyed through the MDI unit for the workpiece coordinate system shift amount and external workpiece zero point offset is loaded into the memory for the workpiece coordinate system shift amount.
3 A write to or read from the workpiece coordinate system shift amount and external workpiece zero point offset with a macro variable is performed using the respective memory.
4 A write to or read from the workpiece coordinate system shift amount and external workpiece zero point offset with the window function is performed using the respective memory.

#2 G92 When the workpiece coordinate system is used (when bit 0 (NWZ) of parameter No. 8136 is 0), if the G command for setting a coordinate system (G92 for M series, G50 for T series (or the G92 command in G command system B or C)) is specified,
0: G command is executed and no alarm is issued.
1: G command is not executed and an alarm PS0010, "IMPROPER G-CODE" is issued.

1220	External workpiece zero point offset value in each axis

[Input type] Setting input
[Data type] Real axis
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This is one of the parameters that give the position of the zero point of workpiece coordinate system (G54 to G59). It gives an offset of the workpiece zero point common to all workpiece coordinate systems. In general, the offset varies depending on the workpiece coordinate systems. The value can be set from the PMC using the external data input function.

1221	Workpiece zero point offset value in workpiece coordinate system 1 (G54)
1222	Workpiece zero point offset value in workpiece coordinate system 2(G55)
1223	Workpiece zero point offset value in workpiece coordinate system 3(G56)
1224	Workpiece zero point offset value in workpiece coordinate system 4 (G57)

1225	Workpiece zero point offset value in workpiece coordinate system 5 (G58)
1226	Workpiece zero point offset value in workpiece coordinate system 6 (G59)

[Input type] Setting input
[Data type] Real axis
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
The workpiece zero point offset values in workpiece coordinate systems 1 to 6 (G54 to G59) are set.

1250	Coordinate system of the reference position used when automatic coordinate system setting is performed

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate system of the reference position on each axis to be used for setting a coordinate system automatically.

	#7	#6	#5	#4	#3	#2	#1	#0
3104					PPD			

[Input type] Parameter input
[Data type] Bit path

#3 PPD Relative position display when a coordinate system is set
0: Not preset
1: Preset

> **NOTE**
> If any of the following is executed when PPD is set to 1, the relative position display is preset to the same value as the absolute position display:
> (1) Manual reference position return
> (2) Coordinate system setting based on G92 (G50 for G code system A on the lathe system)
> (3) Workpiece coordinate system presetting based on G92.1 (G50.3 for G code system A on the lathe system)
> (4) When a T code for the lathe system is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
3290				IWZ	WZO			

[Input type] Parameter input
[Data type] Bit path

#3 WZO Setting a workpiece zero point offset value and workpiece shift value (T series) by MDI key input is:
0: Not disabled.

1: Disabled.

#4 IWZ Setting a workpiece zero point offset value or workpiece shift value (T series) by MDI key input in the automatic operation activation or halt state is:
0: Not disabled.
1: Disabled.

1.5.2.6 Direct input of coordinate system shift

T

Explanation

If the tool position assumed at programming is inputted when the coordinate system actually set is different from the workpiece coordinate system assumed at programming, shift length of the coordinate system actually set and the workpiece coordinate system assumed at programming is set in the workpiece coordinate system shift value.
The procedure leading up to inputting coordinate system position depend on the setting made for bit 2 (PRC) of parameter No.5005.
The procedure in which the coordinate system position of workpiece end face is specified is as follows.

When bit 2 (PRC) of parameter No.5005 is set to 0

1 Cut the workpiece end face tentatively.
2 Display the workpiece coordinate system shift amount setting screen with touching tool to the workpiece end face.
3 Move the cursor to Z axis of MEASUREMENT.
4 Input the workpiece end face position on workpiece coordinate system assumed at programming and press soft key [INPUT].

When bit 2 (PRC) of parameter No.5005 is set to 1

1 Cut the workpiece end face tentatively.
2 Set the position record signal PRC to "1" with touching tool to the workpiece end face.
3 Remove a tool from the workpiece end face.
4 Display the workpiece coordinate system shift amount setting screen.
5 Move the cursor to Z axis of MEASUREMENT.
6 Input the workpiece end face position on workpiece coordinate system assumed at programming and press soft key [INPUT].

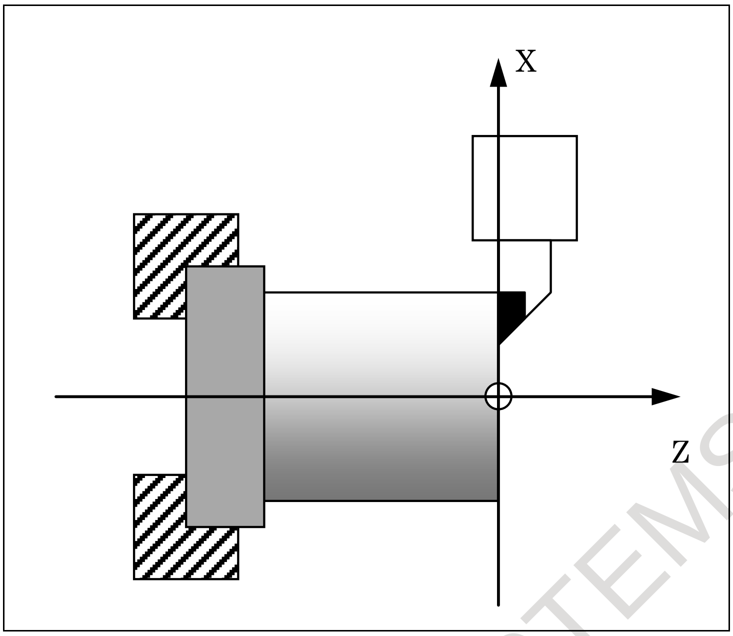


Fig. 1.5.2.6 (a) Direct input of coordinate system shift

Relationship between the input of measurement value and the diameter/radius specification

Whether the measured value input and set value use a diameter value or radius value on the setting of bit 3 (DIAx) of parameter No.1006.

Signal

Position record signal PRC <Gn040.6>

[Classification] Input signal

[Function] This signal is prepared for the function of direct input of tool offset value measured or direct input of coordinate system shift.
It is used to store the data on the positions of the tool for tentative cutting in the control unit. After measuring the workpiece position, input the measured value by the specified manual operation. The difference of the coordinate system actually set and the workpiece coordinate system assumed at programming is set in the workpiece coordinate system shift value.

[Operation] The control unit stores the current position along X and Z axes when the signal turns to "1".

> ⚠ **CAUTION**
> To use this signal, set bit 2 (PRC) of parameter No.5005 to 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn040		PRC						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1201		NWS						

[Input type] Parameter input

[Data type] Bit path

#6 NWS The workpiece coordinate system shift amount setting screen is:
0: Displayed
1: Not displayed

> **NOTE**
> When the workpiece coordinate system shift amount setting screen is not displayed, a workpiece coordinate system shift amount modification using G10P0 cannot be made. If G10P0 is commanded, alarm PS0010, "IMPROPER G-CODE" is issued.

3134	Data display order of each axis on the workpiece coordinate system setting screen and workpiece coordinate system shift amount setting screen

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to Number of controlled axes

Set the data display order of each axis on the workpiece coordinate system setting screen (M series/T series) and workpiece coordinate system shift amount setting screen (T series).
No data is displayed for an axis with 0 set in this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3290				IWZ	WZO			
				IWZ	WZO			

[Input type] Parameter input
[Data type] Bit path

#3 WZO Setting a workpiece zero point offset value and workpiece shift value (T series) by MDI key input is:
0: Not disabled.
1: Disabled.

#4 IWZ Setting a workpiece zero point offset value or workpiece shift value (T series) by MDI key input in the automatic operation activation or halt state is:
0: Not disabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5005						PRC		

[Input type] Parameter input
[Data type] Bit path

#2 PRC For direct input of a tool offset value or workpiece coordinate system shift amount:
0: The position record signal PRC <Gn040.6> is not used.
1: The position record signal PRC <Gn040.6> is used.

1.5.2.7 Each axis workpiece coordinate system preset signals

Overview

The each axis workpiece coordinate system preset signals are function for presetting a workpiece coordinate system shifted due to manual intervention, a machine lock, etc. to a workpiece coordinate system offset from the pre-shift machine zero point by a workpiece origin offset value, using an input signal.
Each axis workpiece coordinate system preset signals become effective by setting bit 6 (WPS) of parameter No. 3006 to 1.
If actual tool length offset need to be considered when workpiece coordinate system preset is done, bit 7 (WTC) of parameter No. 1205 need to be set to 1.

Signal

Each axis workpiece coordinate system preset signals
WPRST1 to WPRST8 <Gn358>

[Classification] Input signal

[Function] These signals are used to preset a workpiece coordinate system shifted due to manual intervention, a machine lock, etc.

[Operation] By changing the signal for the axis for which to perform a workpiece coordinate system preset from "0" to "1", the workpiece coordinate system preset is performed. This cancels the shift amount for the workpiece coordinate system that is due to any of the items below, so that the workpiece coordinate system can be preset to a workpiece coordinate system preset from the pre-shift machine zero point by the workpiece origin offset value.

(a) Manual intervention performed when the manual absolute signal is off
(b) Move command executed in the machine lock state
(c) Movement by handle interruption
(d) Operation using the mirror image function
(e) Setting of a local coordinate system with G52 or shift of a workpiece coordinate system with G92/G50 (for the T series)

NOTE
1 If performing workpiece coordinate system preset with an each axis workpiece coordinate system preset signal during automatic operation, specify it with the M code set in parameters Nos. 11275 and 11276 or perform it during a single block stop.
2 If preset is to be performed with an each axis workpiece coordinate system preset signal during automatic operation, the state must be that in which all axes on the path including the axis on which to perform each axis workpiece coordinate system preset are stopped and no commands are in execution. If not all axes are stopped or if a command is in execution, alarm PS1820, "ILLEGAL DI SIGNAL STATE" is issued.

NOTE

3 When an M code for performing each axis workpiece coordinate system preset is issued during automatic operation, if the corresponding each axis workpiece coordinate system preset signal is not set to 1, alarm PS1820 is issued. It is possible to suppress the alarm by setting bit 0 (WPA) of parameter No. 11277 to 1. In this case, axis workpiece coordinate system preset is not performed.

4 During a reset (RST<Fn001.1> = "1"), preset with an each axis workpiece coordinate system preset signal is disabled. Preset is performed at the point the reset is canceled.

5 During an auxiliary function lock, this function is disabled.

6 In general, if preset is to be performed with an each axis workpiece coordinate system preset signal, the state must be that in which all axes on the path including the axis on which to perform each axis workpiece coordinate system preset are stopped. However, when bit 1 (EAX) of parameter No. 10410 is set to 1 and the axis moves by the PMC axis control, it is possible to exclude the axis from this condition. That is, the preset can be performed without stopping the axis that moves by the PMC axis control. However, when you move the axis that moved by the PMC axis control by the NC commands, it is necessary to stop the axis, and to reflect the amount of movement by PMC axis control to the program coordinate system by the manual return to the reference position etc.

7 Be sure to specify an M code used to turn on each axis workpiece coordinate system preset signal in an independent block.

Each axis workpiece coordinate system preset completion signals WPSF1 to WPSF8<Fn358>

[Classification] Output signal

[Function] These signals notify the PMC of the each axis workpiece coordinate system preset status.

[Output cond.] These signals become "1" in the following case:

- When workpiece coordinate system preset is completed with the corresponding each axis workpiece coordinate system preset signals.

They become "0" in the following cases:

- When the corresponding each axis workpiece coordinate system preset signals change from "1" to "0".
- When a reset is performed.

- Timing chart

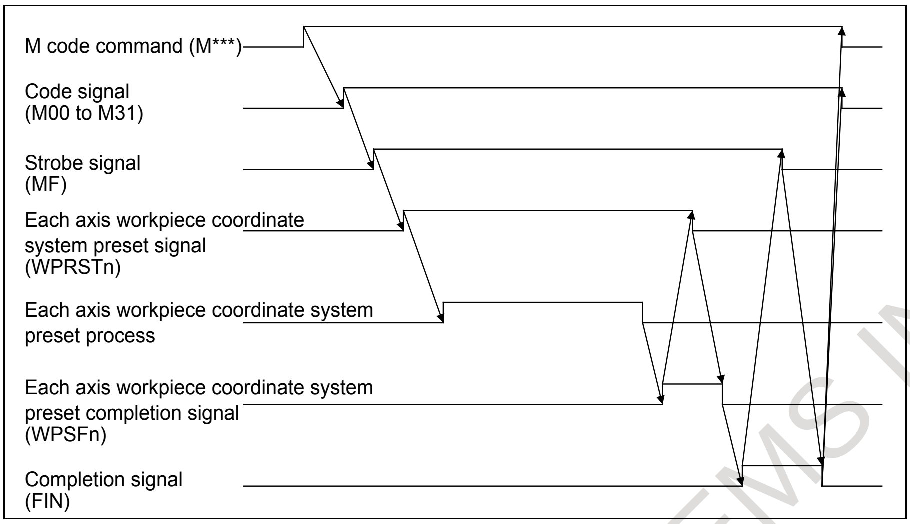


Fig. 1.5.2.7 (a)

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn358	WPRST8	WPRST7	WPRST6	WPRST5	WPRST4	WPRST3	WPRST2	WPRST1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn358	WPSF8	WPSF7	WPSF6	WPSF5	WPSF4	WPSF3	WPSF2	WPSF1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1205	WTC							

[Input type] Parameter input
[Data type] Bit path

#7 WTC When workpiece coordinate system preset is done, actual tool length offset is:
0: Not considered.
1: Considered.

When this parameter is set 1, it is possible to preset the workpiece coordinate system by G-code, MDI operation or the workpiece coordinate system preset signal without canceling the tool length compensation modes.
The compensation vector is kept as the below figure when the workpiece coordinate system preset is done to the coordinate shifted by amount of movement during manual intervention.

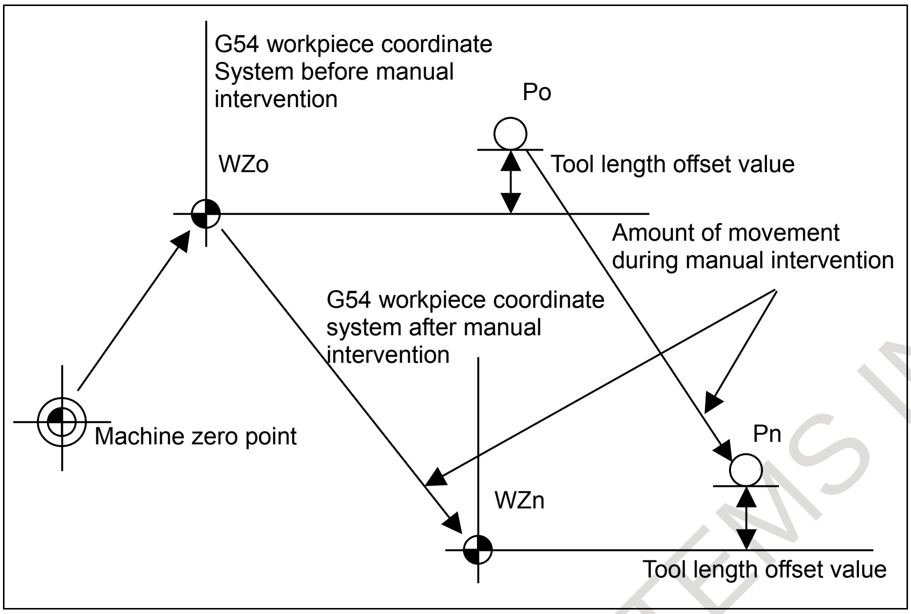


Fig. 1.5.2.7 (b)

	#7	#6	#5	#4	#3	#2	#1	#0
3006		WPS						

[Input type] Parameter input
[Data type] Bit

#6 WPS Each axis workpiece coordinate system preset signal:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10410							EAX	

[Input type] Parameter input
[Data type] Bit axis

#1 EAX When other axis is preset with the workpiece coordinate system preset signal while this axis is moving under the PMC axis control:
0: The alarm PS1820, “ILLEGAL DI SIGNAL STATE” occurs.
1: An alarm doesn’t occur.

11275	The top number of M code used to turn on each axis workpiece coordinate system preset signal

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 999999999

Specify the top number of M code for turning “1” each axis workpiece coordinate system preset signal <Gn358.*> during automatic operation.
When the specified M codes are within the range specified with this parameter and parameter No.11276, each axis workpiece coordinate system preset signal is checked and preset workpiece coordinate system for axis that the signal is turned “1”.
The specified M codes prevent buffering.

> **NOTE**
> When each axis workpiece coordinate system preset signals are turned “1“ more than two signals by an M code, please turn “1“ the signals of all axis at the same timing. If the timing is different, only the axis of the first signal turned “1“ is preset.
> If you want to turn “1“ the signals at the different timing, please specify M code separately.

11276	The number of M code used to turn on each axis workpiece coordinate system preset signal

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 999

Specify the number of M code for turning “1” each axis workpiece coordinate system preset signal <Gn358.*> during automatic operation.
For example, when parameter No.11275 = 100 and parameter No.11276 = 10 are set, From M100 to M109 are used for turning “1” each axis workpiece coordinate system preset signal.
When 0 is set, the number of M code is assumed to be 1.

> **NOTE**
> Set only M code that is not used for another function.
> (M00 to 05, 30, 98, 99, M code used to call the subprogram, etc.)

	#7	#6	#5	#4	#3	#2	#1	#0
11277								WPA

[Input type] Parameter input
[Data type] Bit path

#0 WPA When an M code for turning on the workpiece coordinate system preset signal for an axis is specified, but the signal is not turned on, or an auxiliary function lock is provided:
0: An alarm PS1820, “ILLEGAL DI SIGNAL STATE” is issued.
1: An alarm is not issued.

When bit 6 (PGS) of parameter No. 3001 is set to 0 (M, S, T, and B codes are not output in the high speed program check mode), if an M code for turning on the workpiece coordinate system preset signal for an axis is specified, the system follows the setting of this parameter.

Alarm and message

Number	Message	Description
PS1820	ILLEGAL DI SIGNAL STATE	1. An each axis workpiece coordinate system preset signal was turned on in the state in which all axes on the path including the axis on which to perform preset with the each axis workpiece coordinate system were not stopped or in which a command was in execution. 2. When an M code for performing preset with an each axis workpiece coordinate system preset signal was specified, the each axis workpiece coordinate system preset signal was not turned on. 3. The auxiliary function lock is enabled. 4. When bit 6 (PGS) of parameter No. 3001 was set to 0 (M, S, T, and B codes are not output in the high speed program check mode), an M code for turning on an each axis workpiece coordinate system preset signal in the high speed program check mode was specified.

Notes

> **NOTE**
> The limitations are the same as those for the workpiece coordinate system preset using a program command (G92.1 or G50.3 (for G code system A (T series) or using MDI operation. Thus, before performing preset with this function, cancel each compensation mode (cutter compensation, tool offset, and tool length compensation). Otherwise, the compensation vector will be canceled. If this occurs, specify each compensation mode again. For details, refer to the OPERATOR'S MANUAL (Common to Lathe System/Machining Center System) (B-64694 EN).

1.5.3 Local Coordinate System

Overview

When a program is created in a workpiece coordinate system, a child workpiece coordinate system can be set for easier programming. Such a child coordinate system is referred to as a local coordinate system.

Format

G52 IP _; Setting the local coordinate system
:
G52 IP 0 ; Canceling the local coordinate system
IP_ : Origin of the local coordinate system

Explanation

By specifying G52 IP_;, a local coordinate system can be set in all the workpiece coordinate systems (G54 to G59). The origin of each local coordinate system is set at the position specified by IP_ in the workpiece coordinate system.
Once a local coordinate system is established, the coordinates in the local coordinate system are used in an axis shift command. The local coordinate system can be changed by specifying the G52 command with the zero point of a new local coordinate system in the workpiece coordinate system.
To cancel the local coordinate system and specify the coordinate value in the workpiece coordinate system, match the zero point of the local coordinate system with that of the workpiece coordinate system.

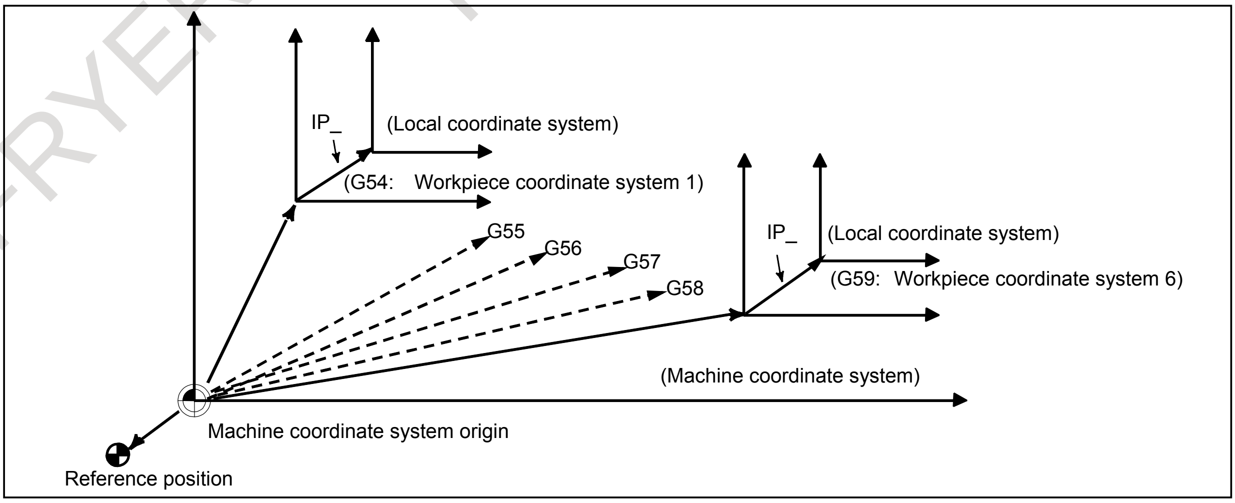


Fig. 1.5.3 (a) Setting the local coordinate system

> ⚠ **CAUTION**
> 1 When bit 2 (ZCL) of parameter No. 1201 is set to 1 and an axis returns to the reference point by the manual reference point return function, the zero point of the local coordinate system of the axis matches that of the work coordinate system.
> The same is true when the following command is issued:
> G52α0;
> α: Axis which returns to the reference point
> 2 The local coordinate system setting does not change the workpiece and machine coordinate systems.
> 3 Whether the local coordinate system is canceled upon reset depends on the specified parameters. The local coordinate system is canceled upon reset when bit 6 (CLR) of parameter No. 3402 or bit 3 (RLC) of parameter No. 1202 is set to 1. In 3-dimensional coordinate conversion mode, however, the local coordinate system is not canceled when bit 2 (D3R) of parameter No. 5400 is set to 1.
> 4 When a workpiece coordinate system is set with the G code command for coordinate system setting (G92 <M series>, G50 <T series> (G92 with G code system B/C for T series)), the local coordinate system is canceled. However, the local coordinate system of an axis for which no coordinate system is specified in a G code block for coordinate system setting remains unchanged.
> 5 G52 cancels the offset temporarily in cutter or tool-nose radius compensation.
> 6 Command a move command immediately after the G52 block in the absolute mode.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1201						ZCL		

[Input type] Parameter input
[Data type] Bit path

#2 ZCL Local coordinate system when the manual reference position return is performed
0: The local coordinate system is not canceled.
1: The local coordinate system is canceled.

> ⚠ **CAUTION**
> ZCL is valid when the workpiece coordinate system is provided (when 0 is set in bit 0 (NWZ) of parameter No. 8136). In order to use the local coordinate system (G52), the workpiece coordinate system option is required.

	#7	#6	#5	#4	#3	#2	#1	#0
1202					RLC			

[Input type] Parameter input
[Data type] Bit path

#3 RLC Local coordinate system is
0: Not cancelled by reset.
1: Cancelled by reset.

> ⚠ **CAUTION**
> 1 When bit 6 (CLR) of parameter No. 3402 is set to 0, and bit 7 (WZR) of parameter No. 1201 is set to 1, the local coordinate system is cancelled, regardless of the setting of this parameter.
> 2 When bit 6 (CLR) of parameter No. 3402 is set to 1, and bit 6 (C14) of parameter No. 3407 is set to 0, the local coordinate system is cancelled, regardless of the setting of this parameter.
> 3 When the 3-dimensional coordinate conversion mode is set, and bit 2 (D3R) of parameter No. 5400 is set to 1, the local coordinate system is not cancelled, regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5400						D3R		

[Input type] Parameter input
[Data type] Bit path

#2 D3R When reset is done by reset operation or reset signal from PMC, 3-dimensional coordinate system conversion mode, tilted working plane command mode and workpiece setting error compensation mode is:
0: Canceled.
1: Not canceled.

1.5.4 Rotary Axis Roll-Over

Overview

The roll-over function of the rotary axis prevents coordinates for the rotary axis from overflowing. The roll-over function of the rotary axis is enabled by setting bit 0 (ROAx) of parameter No. 1008 to 1.
For an incremental command, the tool moves the angle specified in the command. For an absolute command, the coordinates after the tool has moved are values rounded by the angle corresponding to one rotation set in parameter No. 1260. The tool moves in the direction in which the final coordinates are closest when bit 1 (RABx) of parameter No. 1008 is set to 0. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when bit 2 (RRLx) of parameter No. 1008 is set to 1.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx
#1 ROSx Setting linear or rotary axis.

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No.3624)
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No.1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No.3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No.3624) (4) Cannot be used with the rotary axis roll-over and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

	#7	#6	#5	#4	#3	#2	#1	#0
1008						RRLx	RABx	ROAx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROAx The rotary axis roll-over is
0: Invalid.
1: Valid.

> **NOTE**
> ROAx specifies the function only for a rotary axis (for which bit 0 (ROTx) of parameter No.1006 is set to 1)

#1 RABx In the absolute commands, the axis rotates in the direction
0: In which the distance to the target is shorter.
1: Specified by the sign of command value.

> **NOTE**
> RABx is valid only when ROAx is 1.

#2 RRLx Relative coordinates are
0: Not rounded by the amount of the shift per one rotation.
1: Rounded by the amount of the shift per one rotation.

> **NOTE**
> 1 RRLx is valid only when ROAx is 1.
> 2 Assign the amount of the shift per one rotation in parameter No.1260.

1260	The shift amount per one rotation of a rotary axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] Deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the shift amount per one rotation of a rotary axis.
For the rotary axis used for cylindrical interpolation, set the standard value.

Note

> **NOTE**
> This function cannot be used together with the indexing function of the index table (machining center system).

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Rotary axis roll-over function

1.6 AXIS SYNCHRONOUS CONTROL

Overview

When a movement is made along one axis by using multiple servo motors as in the case of a large gantry machine, a command for one axis can drive the multiple motors by synchronizing one motor with the other. An axis used as the reference for axis synchronous control is called a master axis (M-axis), and an axis along which a movement is made in synchronism with the master axis is called a slave axis (S-axis).

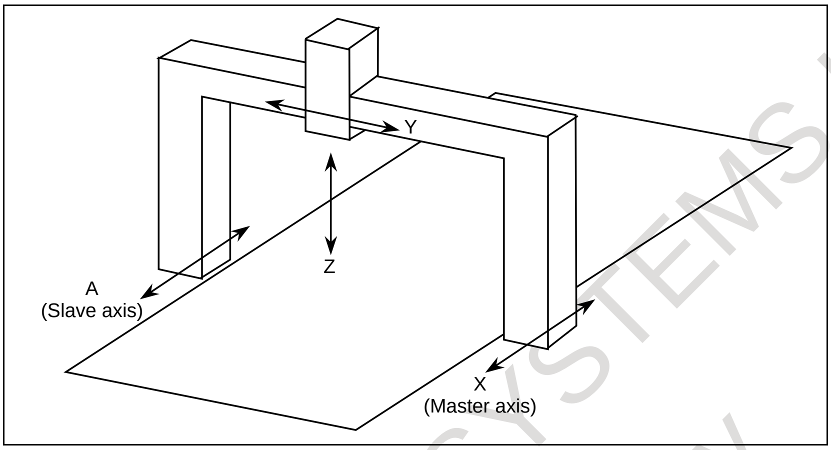


Fig. 1.6 (a) Example of machine with X and A being synchronous axes

Upon power-up or after emergency stop cancellation, the machine position on the slave with that on the master axis can be adjusted by the synchronization establishment function. And a synchronization error amount of the master axis and slave axis is monitored. If the error amount exceeds a certain limit, an alarm is issued and the movement along the axis can be stopped.

1.6.1 Example of Usage

Explanation

- Function to drive multiple axes by command of one axis

In the function to drive multiple axes by command of one axis, there are other functions except the axis synchronous control. The axis synchronous control might be unsuitable depending on the machine. Please select the proper function referring to Table 1.6.1 (a) and Fig. 1.6.1 (a).

Table 1.6.1 (a) Function to drive multiple axes by command of one axis

Function name	Functional overview
Axis synchronous control	• Multiple axes in same path can be drive in synchronism. • Synchronization establishment, Synchronization error check, and Synchronization error compensation can be used. • This function can be used on the large gantry machine. The slave axis can be driven with stability.
Synchronous control	• Multiple axes in all paths can be drive in synchronism. • Parking can be used. • This function can be used on the combined machine, etc.
Tandem control	• Two motors in same path can be drive one axis. • Position is controlled only for the master axis. The slave axis is used only to output torque same as master axis. • This function can be used as follows. Two motors drive one ball screw. Motor having two windings is driven.

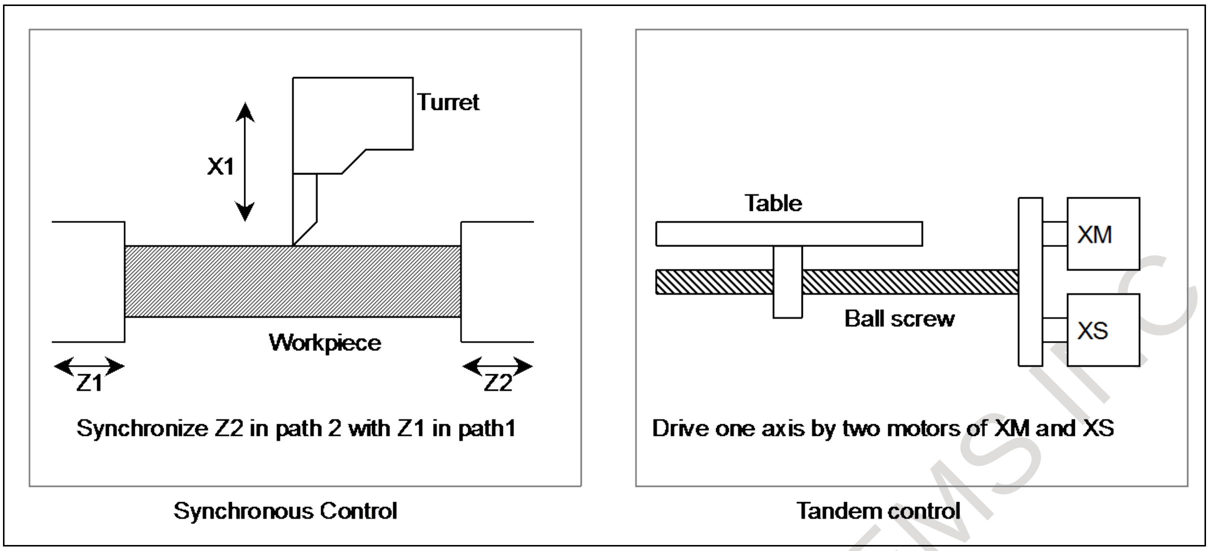


Fig. 1.6.1 (a) Application example

- Axis synchronous control

The axis synchronous control can drive multiple motors in synchronism by the command for one axis. The functions specific to axis synchronous control are as listed below.

Table1.6.1 (b) Functions of axis synchronous control

Function name	Overview
Automatic Setting for Grid Position Matching	The reference counters (grid positions) difference between the master axis and slave axis can be automatically set to a parameter. The CNC automatically adjusts reference positions (grid positions) of the master axis and the slave axis.
Synchronization establishment	Upon power-up or after emergency stop cancellation, the machine position on the slave axis with that on the master axis can be adjusted.
Synchronization error check	The synchronization error amount is monitored. If the error amount exceeds a certain limit, an alarm is issued and the movement along the axis can be stopped.
Axis Synchronous Control Torque Difference Alarm	The torque command difference between the master axis and the slave axis is observed. If the difference exceeds certain limit, a servo alarm can be issued.
Synchronous axes automatic compensation	If there is the torque difference according to a mechanical error (the pitch error of the scale, the expansion difference of the ball screw, etc.) between the master axis and the slave axis, servo control can automatically compensate the position of the slave axis to decrease torque difference. Overheat of the motor can be prevented when the conflicting torque is output from a master axis and a slave axis. (Note)
Integrator copy function	The main component of the torque command is copied from a master axis to a slave axis, and it shares. This function is similar to the tandem control. However, a movement that is more stable than the tandem control can be expected. Because the velocity damping control is executed on each motor. In case that the position control is executed on two motors independently, when the torque commands have the offset by the interference between axes, this function works appropriately. (Note)
Synchronization error compensation	When the synchronization error of the positional deviation value is larger than the parameter value, compensation pulses can be output to a slave axis for decreasing the synchronization error. If the positional deviation corresponding to the load were generated like a past analog servo, this function was effective for decreasing the synchronization error. When a current digital servo is applied, positional deviation corresponding to the load are not generated. Therefore, the function need not be applied.

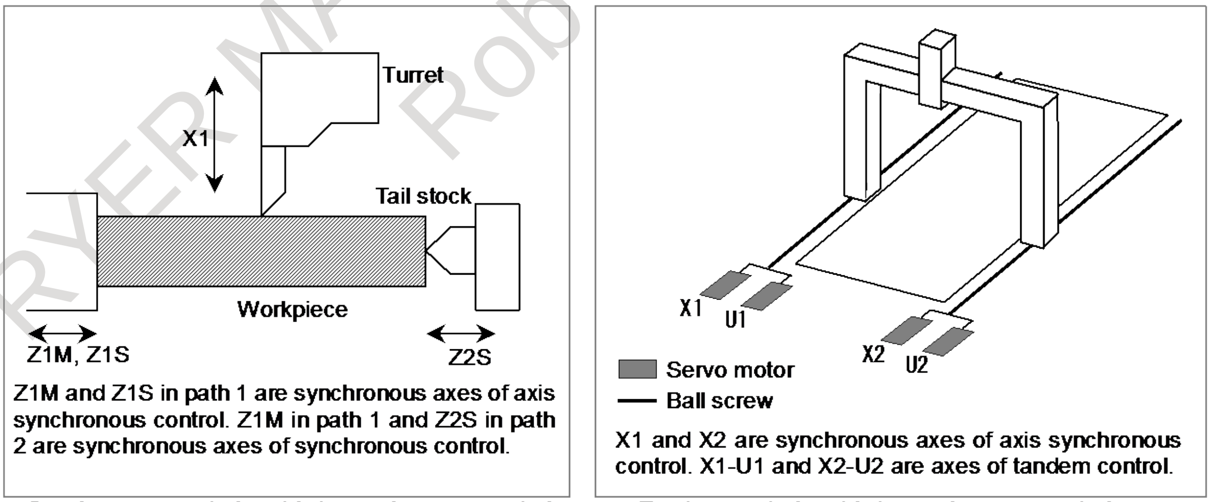
> **NOTE**
> This function is servo function. For details of this function, refer to "FANUC AC SERVO MOTOR α*i*/β*i* series, LINEAR MOTOR LiS series, SYNCHRONOUS BUILT-IN SERVO MOTOR DiS series PARAMETER MANUAL (B-65270EN)".

- Use with other functions

There are the functions to drive multiple axes by command of one axis, and the axis synchronous control can be used together with the following functions.

Table 1.6.1 (c) Use with other functions

Function name	Use with axis synchronous control
Synchronous control	The master axis of the axis synchronous control can be also set to the master axis of the synchronous Control. (Example) Path 1 X1M (Master of axis synchronous control and synchronous control) X1S (Slave of axis synchronous control) Path 2 X2S (Slave of synchronous control)
Tandem control	Tandem control can be used with each of the master axis and the slave axis of axis synchronous control respectively. The same restriction on axis arrangement as imposed in the case of normal tandem control is imposed. No particular restriction is imposed on axis synchronous control.
Flexible synchronization control	The slave axis of flexible synchronization control can be set to the master axis of axis synchronous control by setting bit 0 (FSS) of parameter No. 8307 to 1. Then, it is necessary to execute beginning and the end of flexible synchronization control while axis synchronous control.
Superimposed control	The slave axis of superimposed control can be set to the master axis of axis synchronous control by setting bit 0 (FSS) of parameter No. 8307 to 1. Then, it is necessary to execute beginning and the end of superimposed control while axis synchronous control. It is necessary to set superimposed ahead signals OVLN <Gn531.4> to "1" during superimposed control is begun.



Synchronous control and Axis synchronous control   Tandem control and Axis synchronous control

Fig. 1.6.1 (b) Use with other functions

When the axis of large gantry machine is driven by two motors, please apply the axis synchronous control. (See the left side of the below Fig. 1.6.1 (c).) The stability and the accuracy increase more than the tandem control because the velocity control is executed in the slave axis.

In the above gantry machine, if two motors drive the one ball screw, please apply the tandem control. (See the right side of the Fig. 1.6.1 (b).)

- Example of using axis synchronous control

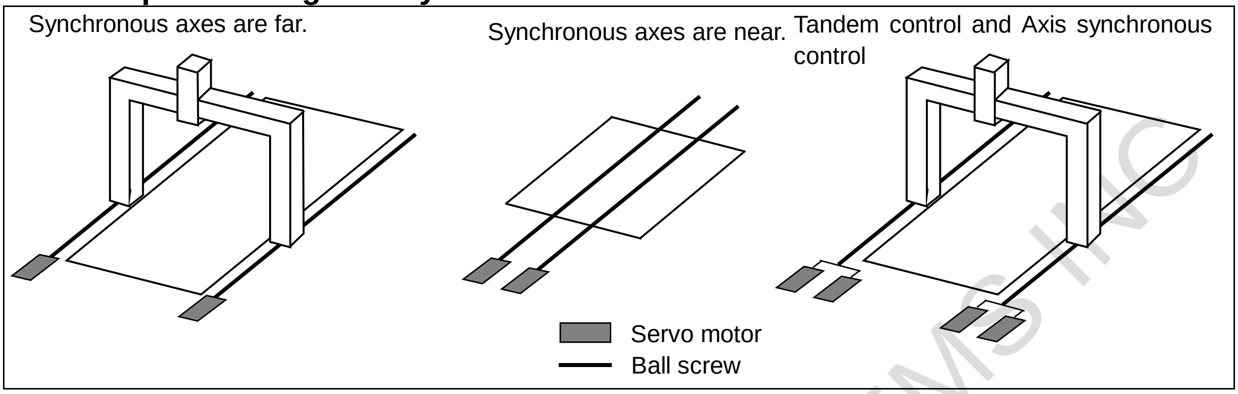


Fig. 1.6.1 (c) Example of using axis synchronous control

If there is the torque difference between a master axis and a slave axis, please apply the synchronous axes automatic compensation or the Integrator copy function.

1.6.2 Procedure to Start-Up

Explanation

The procedure to start-up the axis synchronous control is as follows.

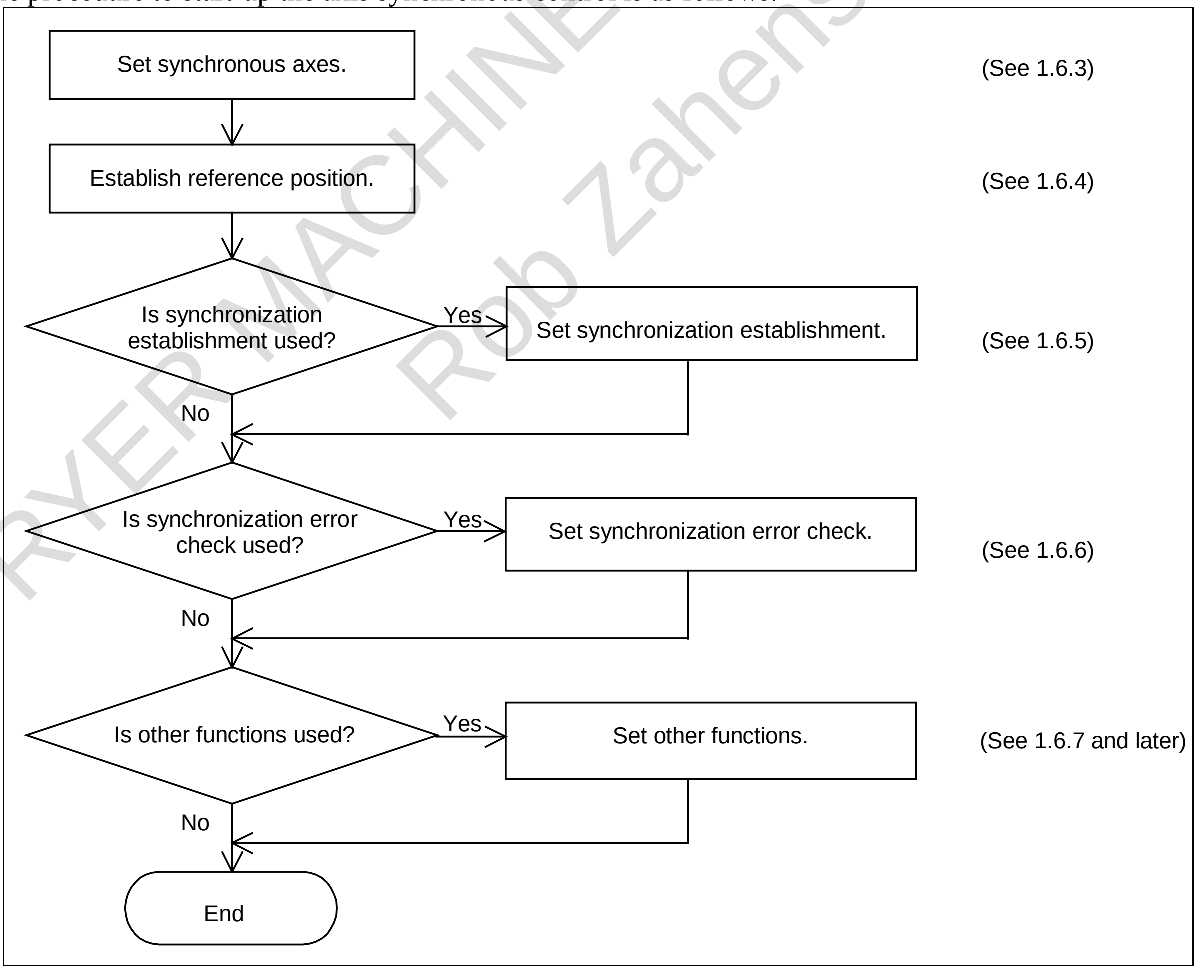


Fig. 1.6.2 (a)

1.6.3 Setting of Synchronous Axes

Explanation

- Master axis and slave axis for axis synchronous control

An axis used as the reference for axis synchronous control is called a master axis (M-axis), and an axis along which a movement is made in synchronism with the master axis is called a slave axis (S-axis).
By setting the axis number of a master axis within a path to the parameter No. 8311 of the slave axis, the axis configuration for axis synchronous control is set. The multiple slave axes can set within the range of the maximum controlled axes of a path. The slave axis cannot be set as the master axis of other synchronous axes.

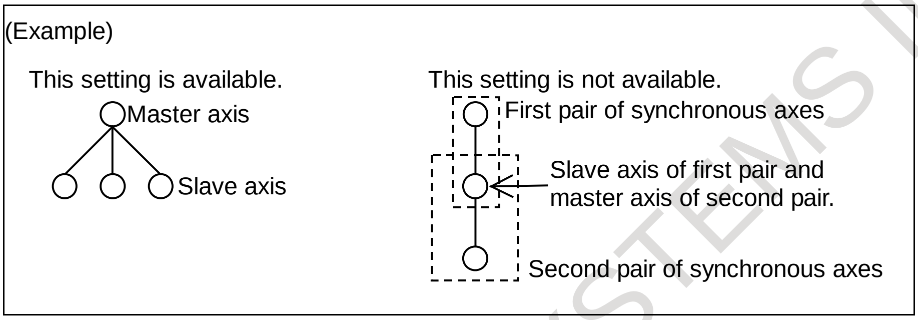


Fig. 1.6.3 (a)

- Maximum pairs of synchronous axes

Maximum pairs of synchronous axes (maximum number of combinations for master axis and slave axis) are 4 pairs.

- Synchronous operation and normal operation

Operation where axis synchronous control is turned on (enabled) to make a movement along the slave axis in synchronism with the master axis is called a synchronous operation. Operation where axis synchronous control is turned off (disabled) to move a master axis and a slave axis independently is called a normal operation.

(Example)

Automatic operation when the master axis is the X-axis and the slave axis is the A-axis:
In synchronous operation, both the X-axis and the A-axis are moved according to the programmed command Xxxxx for the master axis.
In normal operation, the master axis and the slave axis are moved independently as in the case of normal CNC control. The programmed command Xxxxx makes a movement along the X-axis. The programmed command Aaaaa makes a movement along the A-axis. The programmed command Xxxxx Aaaaa makes movements along the X-axis and A-axis at the same time.

The mode of operation can be switched between synchronous operation and normal operation by an input signal, or synchronous operation can be performed at all times. Which mode to use can be set using bit 5 (SCAx) of parameter No. 8304.

Switching between synchronous operation and normal operation by using an input signal

When bit 5 (SCAx) of parameter No. 8304 is set to 0 for the slave axis, the signals for selecting the axis for axis synchronous control SYNC1 to SYNC8<Gn138> / signals for selecting the manual feed axis for axis synchronous control SYNCJ1 to SYNCJ8<Gn140> is used to switch between synchronous operation and normal operation. When SYNC/SYNCJ = 1, synchronous operation is selected. When SYNC/SYNCJ = 0, normal operation is selected.
Synchronization error compensation cannot be used when the mode of operation is switched between synchronous operation and normal operation.

> ⚠ **CAUTION**
> To switch between synchronous operation and normal operation, command the relevant M code (set to the parameter No. 8337 or 8338).

The synchronous operation and the normal operation can be changed by setting each signal SYNC / SYNCJ by each operation of automatic operation and manual operation.

Setting for using synchronous operation at all times

When bit 5 (SCAx) of parameter No. 8304 for the slave axis is set to 1, synchronous operation is performed at all times, regardless of the setting of the signal SYNC/SYNCJ.

- Axis synchronous control status signals

During axis synchronous control, the axis synchronous control status signals SYNO1 to SYNO8<Fn532> becomes 1. However, this signal might not be the same as the status of signal SYNC / SYNCJ or bit 5 (SCAx) of parameter No. 8304. For instance, this signal is “0” during emergency stop, servo alarm, servo off, and controlled axes detach.

- Synchronous control axis name

There is no problem whether the name of a master axis and the name of a slave axis is the same or not.

Restrictions on using the same name for the master axis and slave axis

If the same axis name is assigned to the master axis and slave axis, manual operation only is allowed in normal operation. Automatic operation cannot be performed.

Axis name subscript and extended axis name

If the axis name of synchronous axis is set by using the axis name subscript or the extended axis name, the axes can be distinguished from each other on the screen display, or which of those axes issued an alarm can be identified.
The axis name subscript is set by the parameter No. 3131, and the extended axis name is set by the bit 0 (EEA) of parameter No.1000, parameter No.1025, and parameter No.1026.

- Current position display screen

On a screen such as the current position display screen, a slave axis is also displayed. The display of a slave axis can be disabled by the following parameter settings.

- Set the last to the slave axis of parameter No.3130 (axis display order for current position display screens).
- Set 1 to bit 0 (NDPx) of parameter No.3115 and bit 1 (NDAx) of parameter No.3115.

Example

In the example below, movements along the X1-axis and X2-axis are made in synchronism with the XM-axis.

Table 1.6.3 (a)

Axis name indication	Controlled axis number	Axis name No. 1020	Subscript No.3131	Master axis number No.8311	Operation
XM	1	88	77	0	
Y	2	89	0	0	
X1	3	88	49	1	A movement is made in synchronism with the XM-axis.
X2	4	88	50	1	A movement is made in synchronism with the XM-axis.

When one master axis has multiple slave axes, synchronization error compensation, synchronization establishment, and synchronization error check are performed for each slave axis independently.

- Slave axis mirror image

By setting parameter No. 8312, a mirror image can be applied to a slave axis placed in synchronous operation. When the mirror image function is enabled, the direction in which the absolute and relative coordinates change is the same as for the machine coordinates.
At this time, synchronization error compensation, synchronization establishment, synchronization error check, and correction mode cannot be used.
The mirror image set by bit 0 (MIRx) of parameter No. 0012 cannot be applied to the slave axis. Because this mirror image differs from the mirror image set by bit 0 (MIRx) of parameter No. 0012, it does not affect mirror image signals MI1 to MI8 <Gn106> or mirror image check signals MMI1 to MMI8 <Fn108>.

- Automatic slave axis parameter setting

Axis synchronous control involves parameters which must be set to the same values for the master axis and slave axis. (See the parameter list in "Parameters which must be set to the same value for the master and slave axes" to appear later. If bit 4 (SYPx) of parameter No. 8303 is 1, setting values for these parameters for the master axis causes the same values to be set to the parameters for the slave axis. For details, see Subsection “Automatic Slave Axis Parameter Setting”.

1.6.4 Reference Position Establishment

1.6.4.1 Procedure of reference position establishment

The procedure of reference position establishment using grid points is as follows.

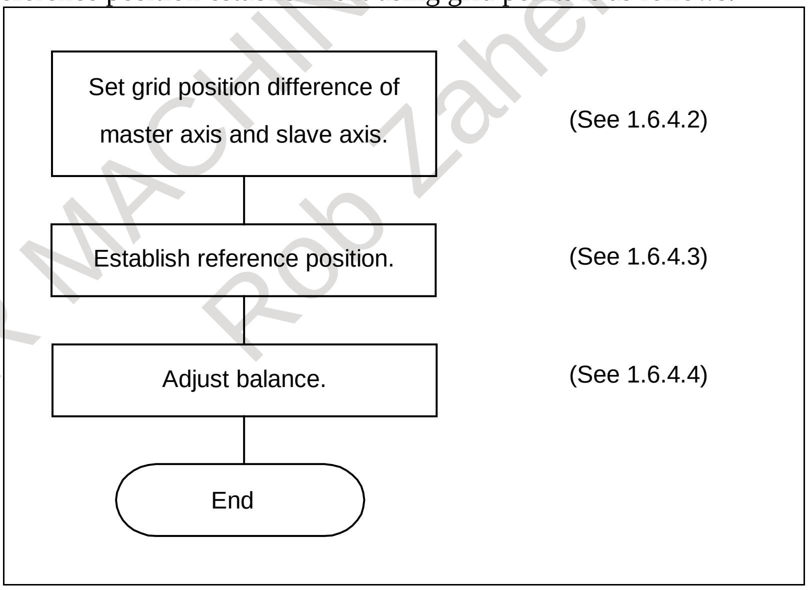


Fig. 1.6.4.1 (a)

- Absolute position detection

When the absolute position detection is used, do not execute the setting the zero point using MDI operation that sets bit 4 (APZ) of parameter No.1815 directly for the re-setting of the reference point establishment in the maintenance work. After setting up the deceleration dog or clarifying the start position for reference position establishment, perform the setting the zero point of the absolute position detector.
By setting bit 7 (SMA) of parameter No. 8302 is set to 1, when the bit 4 (APZx) of parameter No. 1815 for an axis in synchronous operation is set to 0, and APZx of the pairing axis in synchronous operation is also set to 0.

- Reference point setting with mechanical stopper

If the reference point setting with mechanical stopper is used, see Subsection “Reference point setting with mechanical stopper”.

- Distance coded linear scale interface, Linear scale with distance-coded reference marks (serial)

If the distance coded linear scale interface or the linear scale with distance-coded reference marks (serial) is used, see Subsection “Distance coded linear scale interface, Linear scale with distance-coded reference marks (serial)”.

- Normal operation

The reference point establishment by the normal operation is the same as a usual axis.

1.6.4.2 Setting of grid position

When manual reference position return operation is performed along axes under axis synchronous control, the machine is placed at the reference position on the master axis and slave axis according to the same sequence as for normal reference position return operation.
The sequence is the same as the grid method for one axis only. However, only the deceleration signal for the master axis is used. When the deceleration signal is set to 0, the machine gradually stops along the master axis and slave axis, and then an FL feedrate is set. When the deceleration signal is set to “1”, the machine moves to a grid position along each of the master axis and slave axis, then stops.

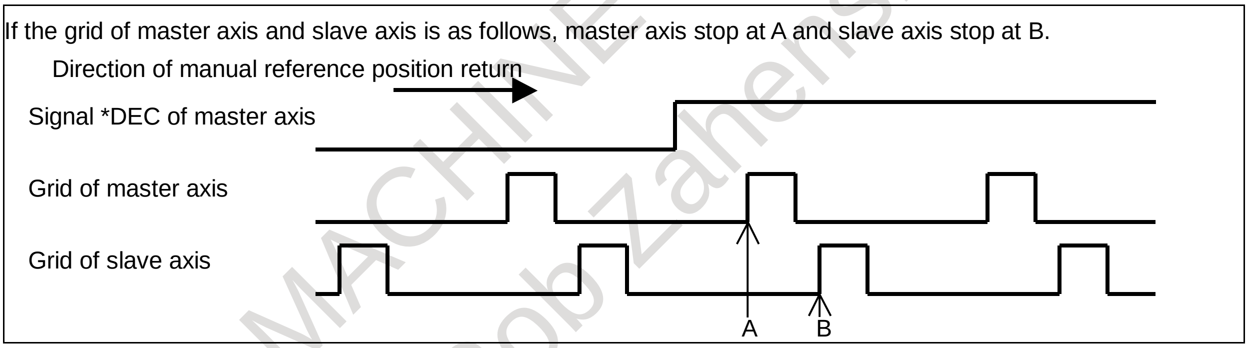
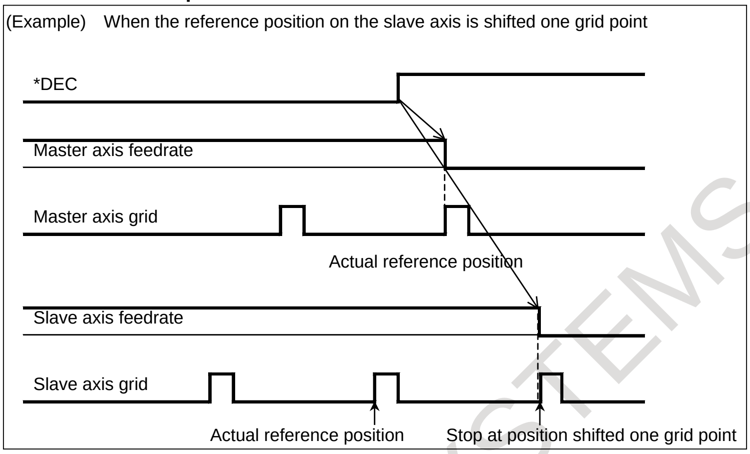


Fig. 1.6.4.2 (a)

NOTE

When the grid position difference between the master axis and slave axis is large, a reference position shift may occur, depending on the timing of the *DEC signal set to “1”. In the example below, the shift along the slave axis is so large that the position shifted one grid point from the actual reference position is regarded as the reference position.



Before axis synchronous control can be performed, the reference position (grid position) on the slave axis must be adjusted to the reference position on the master axis.
If the grid position difference of the master axis and the slave axis is set to the parameter by either of the following methods, the master axis and the slave axis can be stopped at the grid position of the master axis.

- Automatic setting for grid position matching
- Reference position shift function

- Automatic setting for grid position matching

The automatic setting for grid position matching can be automatically set the reference counters (grid positions) difference in parameter No.8326, and the CNC automatically adjusts the reference positions (grid positions) of the master axis and slave axis. The operation procedure is as follows.

1. Select the synchronous operation.
2. Set the following parameters.

No.1844=0	: Distance to the first grid point	(NOTE 1)
No.1850=0	: Amount of grid shift and reference position shift	(NOTE 1)
Bit4 (SFDx) of No.1008 =0	: Reference position shift functions is disabled.	(NOTE 1)
Bit0 (ATEx) of No.8303 =1	: Automatic setting for grid positioning is enabled.	(NOTE 2)
Bit1 (ATSx) of No.8303 =1	: Automatic setting for grid positioning is started.	(NOTE 2)

   If bit 5 (APCx) of No.1815 =1, bit 4 (APZx) of No.1815 =0
3. Turn off the power then turn on the power.
4. Set the REF mode (or JOG mode in the case of reference position setting without dogs), and make movements in the reference position return direction along the master axis and slave axis.
5. The movements along the master axis and slave axis automatically stop, and a grid difference value is set in parameter No. 8326. At this time, bit 1 (ATSx) of parameter No. 8303 is set to 0, and the power-off request alarm PW0000, “POWER MUST BE OFF” is issued. Turn off the power then turn on the power again.

> **NOTE**
> 1 This parameter is set to the all synchronous axes.
> 2 This parameter is set to the slave axis.
> 3 Parameter setting
> When bit 1 (ATSx) of parameter No. 8303 is set, bit 4 (APZx) of parameter No. 1815 and parameter No. 8326 for the master axis and slave axis are set to 0. When the operator sets parameter No. 8326 (MDI, G10L50), bit 0 (ATEx) of parameter No. 8303 is set to 0.
> 4 This function cannot be used together with the reference position shift function.

- Reference position shift function

If the reference position shift function is used, set the following parameters after the parameter No.8326 is set automatically by the operation procedure of the automatic setting for grid position matching.

Bit4 (SFDx) of No.1008 = 1 : Reference position shift function is enabled. (NOTE 1)
No.1850 = Amount of parameter No.8326: Amount of reference position shift (NOTE 2)
No.8326 = 0 : Reference counters difference between synchronous axes (NOTE 2)

> **NOTE**
> 1 This parameter is set to the all synchronous axes.
> 2 This parameter is set to the slave axis.

1.6.4.3 Reference position establishment

The procedure of reference position establishment is as follows.

1. Select the synchronous operation.
2. When the reference position is shifted, add the shift amount to parameter No.1850 (Amount of grid shift and reference position shift) of the all synchronous axes.
3. If bit 5 (APCx) of parameter No.1815 is set to 1, set 0 to bit4 (APZx) of parameter No.1815.
4. Turn off the power then turn on the power.
5. Perform the manual reference position return.

- Reference position return operation of low-speed type

When bit 4 (SLR) of parameter No. 8305 is set to 1, if G28 is specified for an axis under axis synchronous control for which the reference position is not established, reference position return operation of low-speed type is performed.

1.6.4.4 Balance adjustment

The synchronous axes should be adjusted the balance because multiple motors are cooperatively driven. The procedure of balance adjustment is as follows.

1. Stop the synchronous axes at one side (See Fig. 1.6.4.4 (a) Measuring position 1), and record the absolute coordinate of slave axis. This coordinate value is assumed the coordinate value A.
2. Terminate synchronous status, and select the normal operation.
3. Move the slave axis by manual handle feed so that the actual current value in the servo tuning screen may become near.
4. Straightness of synchronous axes is adjusted if necessary. Adjust the position of slave axis by manual handle feed while confirming the measured straightness and the actual current value in the servo tuning screen. The aim of actual current value in the servo tuning screen is 5%-10% of the rated current value.
5. Select the synchronous operation. The following procedures are performed by synchronous operation.

6. Record the absolute coordinate of slave axis. This coordinate value is assumed the coordinate value B. (See Fig. 1.6.4.4 (b)) Add the value of “coordinate value B - coordinate value A” to parameter No.1850 of slave axis.
7. If bit 5 (APCx) of parameter No.1815 is set to 1, set 0 to bit4 (APZx) of parameter No.1815.
8. Turn off the power then turn on the power.
9. Perform the manual reference position return.
10. Stop the synchronous axes at other side (See Fig. 1.6.4.4 (a) Measuring position 2), and confirm the actual current value in the servo tuning screen. If the value is different, adjust the position of the slave axis by using the pitch error compensation, the straightness compensation, and the external machine coordinate system shift, etc.
11. Move the synchronous axes to the center (See Fig. 1.6.4.4 (a) Measuring position 3), and confirm the actual current value in the servo tuning screen. If the value is different, adjust the position of the slave axis by using the pitch error compensation, the straightness compensation, and the external machine coordinate system shift, etc.

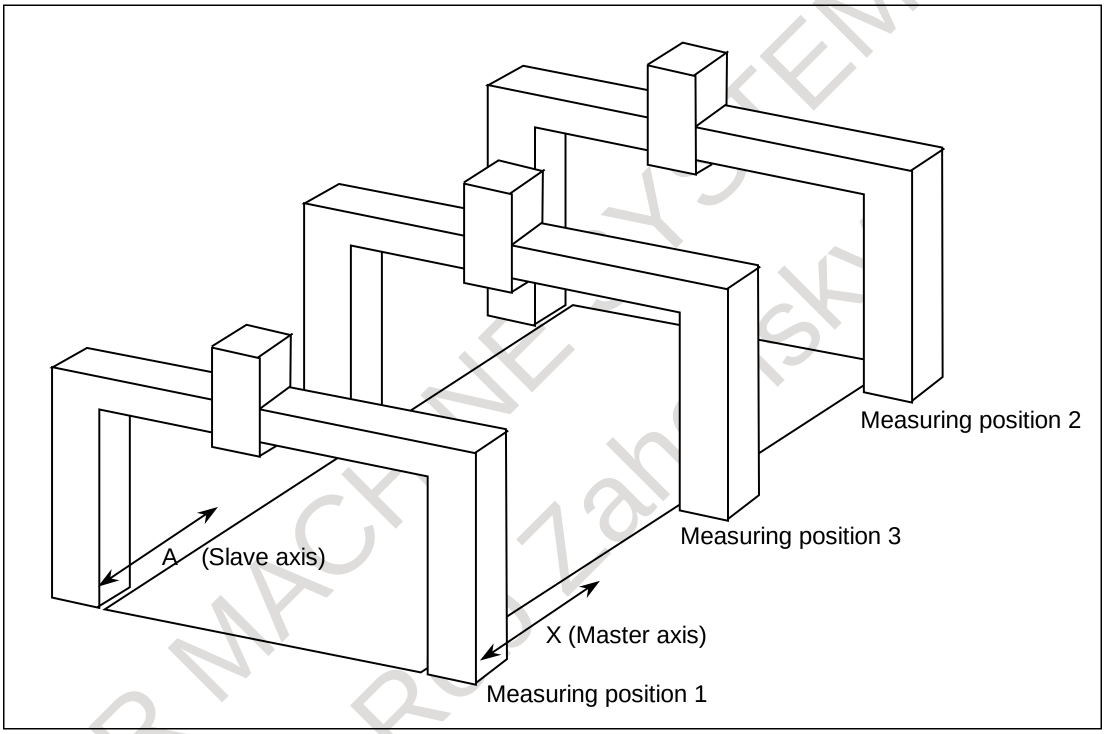


Fig. 1.6.4.4 (a) Measuring position

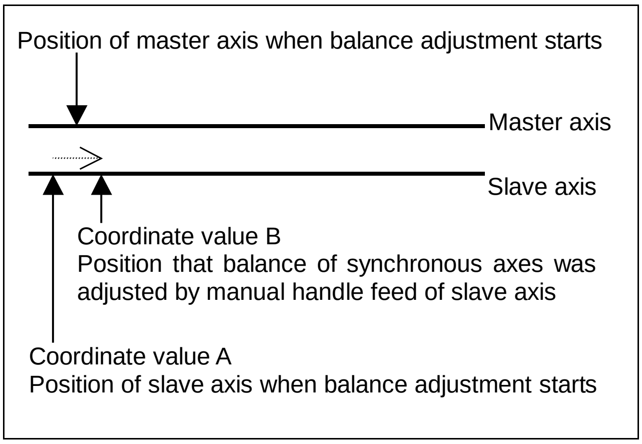


Fig. 1.6.4.4 (b) Position of balance adjustment

- Pitch error compensation and straightness compensation

The pitch error compensation and the straightness compensation are executed independently on the master axis and the slave axis.

- External machine coordinate system shift

By setting bit 7 (SYEx) of parameter No. 8304 to 1 to the slave axis, the slave axis can be shifted by the same amount as specified for the master axis when external machine coordinate system shift is specified by external data input/output for the master axis in synchronous control. When bit 7 (SYEx) of parameter No. 8304 is set to 0, the external machine coordinate system shift is executed independently on the master axis and the slave axis.

1.6.4.5 Maintenance

When motor is exchanged, perform the recovery operation from setting of grid position.
When the parameter is lost by an unexpected situation, restore the backup data of the parameter and perform only reference position establishment again. Thus, the reference position becomes the same position of last time.

1.6.4.6 Reference point setting with mechanical stopper

The procedure of reference position establishment by the reference point setting with mechanical stopper is as follows.

1. If the balance adjustment is necessary, execute the balance adjustment by the procedure 2-4 in Subsection "Balance adjustment". If the balance adjustment is completed, select the synchronous operation.
2. Perform the reference position establishment by the reference point setting with mechanical stopper.

1.6.4.7 Distance coded linear scale interface and linear scale with distance-coded reference marks (serial)

The procedure of reference position establishment by the distance coded linear scale interface or the linear scale with distance-coded reference marks (serial) is as follows.

1. If the balance adjustment is necessary, execute the balance adjustment by the procedure 2-4 in Subsection "Balance adjustment". If the balance adjustment is completed, select the synchronous operation.
2. Perform the reference position establishment by the distance coded linear scale interface or the linear scale with distance-coded reference marks (serial).
3. If the reference position of the master axis and the slave axis shifts, add the shift amount to parameter No.1883 and No.1884. Thus, the reference position of the master axis and slave axis becomes the same position.

NOTE

1 It is also possible to use the master axis or the slave axis in the opposite direction, in the distance coded linear scale interface and the linear scale with distance-coded reference marks (serial).

2 To use the distance coded linear scale interface and the linear scale with distance-coded reference marks (serial), the option "Linear scale I/F with absolute address reference mark" is required.

1.6.5 Synchronization Establishment

Explanation

Upon power-up or after emergency stop cancellation, the machine positions on the master axis and slave axis under axis synchronous control are not always the same. In such a case, the synchronization establishment function adjusts the machine position on the slave to that on the master axis.

- Synchronization establishment method

The method of synchronization establishment differs whether the synchronization error compensation is performed or not. For the synchronization establishment when synchronous error compensation is performed, see Subsection “Synchronization Error Compensation”.

- Synchronization establishment based on machine coordinates

To perform synchronization establishment, enable synchronization establishment based on machine coordinates by setting bit 7 (SOFx) of parameter No. 8303 to 1.
This synchronization establishment is performed by outputting the machine coordinate difference between the master axis and slave axis as command pulses for the slave axis. Thus, the position of the master axis and the slave axis become the same. A maximum allowable compensation value to be used for synchronization establishment is the value in parameter No. 8325. As a maximum allowable compensation value, set a maximum allowable value by which the machine may move abruptly.
If a compensation value is larger than the value set in this parameter, an alarm SV0001, “SYNC ALIGNMENT ERR” is issued, and synchronization establishment is not performed. Moreover, when parameter No. 8325 is set to 0, synchronization establishment is not performed.
The result of comparing the positional difference between the master axis and slave axis with a maximum allowable compensation value for synchronization establishment can be checked using the synchronization establishment enable state output signals SYNOF1 to SYNOF8 <Fn211>.

- First synchronization establishment after power-up

In the first synchronization establishment after power-up, there are a method by the manual reference position return and other methods.

Synchronization establishment based on manual reference position return operation

When manual reference position return operation is performed along axes under axis synchronous control, the machine is placed at the reference position on the master axis and slave axis according to the same sequence as for normal reference position return operation. Therefore, when the grid position of the master axis and the slave axis shifts and the shift amount is not set to parameter No.8326, the master axis and the slave axis stop at each grid position. For details, see Subsection “Setting of grid position”.

Synchronization establishment except manual reference position return

In case of synchronization establishment by absolute position detection or except manual reference position return, when the reference position is established, the synchronization establishment is performed automatically.
When the position is set by the following functions, the synchronization establishment is executed.
- Temporary absolute coordinate setting
- Checking the stored stroke during the time from power–on to the reference position establishment

- Synchronization establishment after emergency stop cancellation, etc.

Synchronization establishment is also performed when servo position control is turned on, for example, at emergency stop cancellation, servo alarm cancellation, or servo-off cancellation .
However, when the controlled axes detach is canceled, the synchronization establishment is not performed until the reference position is established by the manual reference position return etc.

- One-direction synchronization establishment

The synchronization establishment can be performed by setting bit 0 (SSO) of parameter No. 8305 to 1 to move the machine in one direction along the master axis and slave axis. The move direction depends on the reference position setting based on bit 0 (SSAx) of parameter No. 8304. When bit 0 (SSAx) of parameter No. 8304 = 0, for example, the machine coordinate on the master axis or slave axis, whichever larger, is used as the reference point. So, the machine moves in the + direction along the axes.
When bit 1 (SSE) of parameter No. 8305 is set to 1, normal synchronization establishment is performed instead of one-direction synchronization establishment after an emergency stop.

- Axis movement in synchronization establishment

A machine coordinate difference between the master axis and slave axis is output at a time as command pulses, when bit 0 (SJR) of parameter No. 8306 is 0. So, if the compensation value is large, the machine abruptly makes a large movement.
When bit 0 (SJR) of parameter No. 8306 = 1 and one-direction synchronization establishment is invalid (bit 0 (SSO) of parameter No. 8305 = 0), the axis movement of the synchronization establishment in the axis synchronous control can be executed with the feedrate of manual rapid traverse and acceleration/deceleration after interpolation in rapid traverse. In the synchronization establishment using this function, the setting values used for manual rapid traverse and acceleration/deceleration after interpolation in rapid traverse are the parameter values set with slave axis.
The followings are the parameters related to this function.

- No.1424 :Manual rapid traverse rate for each axis (When 0 is set, the rate set in parameter No. 1420)
- No.1620 :Time constant used for linear acceleration/deceleration for each axis
- No.1621 :Time constant T2 used for bell-shaped acc./dec. in rapid traverse for each axis
- No.1401#1 :Positioning (G00) is performed with non-linear or linear interpolation
- No.1603#4 :Acceleration/deceleration used for positioning of linear interpolation type is acceleration fixed or time fixed type

⚠WARNING

1 Do not execute automatic or manual operation while the axes are moving for the synchronization establishment.
2 Do not use other function while the axes are moving for the synchronization establishment.
3 Do not change operation mode while the axes are moving for the synchronization establishment.

NOTE

1 For the axes engaged in the synchronization establishment, the signals below are invalid:
- Rapid traverse override
- External deceleration
- Dry run
- Interlock

2 When the one-direction synchronization establishment function under axis synchronous control is valid (bit 0 (SSO) of parameter No.8305 is set to 1), this function is disabled, regardless of bit 0 (SJR) of parameter No. 8306

3 When the reference position has not been established, the synchronization establishment based on machine coordinates is not executed.

4 When the stored stroke limit check immediately after power-on is enabled (Bit 0 (DOT) of parameter No. 1311 = 1), the synchronization establishment based on machine coordinates is executed as follows.
- Bit 6 (LZR) of parameter No. 1300 = 0 : It is executed after power-on.
- Bit 6 (LZR) of parameter No. 1300 = 1 : It is executed after reference position establishment.

1.6.6 Synchronization Error Check

1.6.6.1 Synchronization error check

A synchronization error amount is monitored at all times. If the error amount exceeds a certain limit, an alarm is issued and the movement along the axis is stopped. For the synchronization error check when synchronous error compensation is performed, see Subsection “Synchronization Error Compensation”.

- Synchronization error check based on machine coordinates

When the value other than 0 is set in parameter No.8314, a synchronization error check based on machine coordinates is enabled.
The machine coordinate on the master axis is compared with that on the slave axis. When the error between the machine coordinates exceeds the value set in parameter No. 8314, the alarm SV0005, “SYNC EXCESS ERR (MCN)” is issued, and the motor is stopped immediately. The alarm SV0005 is output to both the master axis and the slave axis.
This synchronization error check can also be made in the emergency stop, servo-off, and servo alarm state.
If a synchronization error check is made when the mode of operation is switched between synchronous operation and normal operation with an input signal, an error check is made even in normal operation. So, even if the axis synchronous control selection signal (SYNC) or the axis synchronous control manual feed selection signal (SYNCJ) is set to “0” by mistake during synchronous operation, damage to the machine can be prevented.
The machine coordinates on the master axis and slave axis can be checked using the machine coordinate match state output signals SYNMT1 to SYNMT8 <Fn210>.

- Synchronization error check based on a positional deviation value

When the value other than 0 is set in parameter No.8323, a synchronization error check based on a positional deviation value is enabled. The servo positional deviation value of the master axis and slave axis is monitored during axis synchronous control. When the positional deviation value exceeds the limit value set in parameter No. 8323, the alarm DS0001, “SYNC EXCESS ERR (POS DEV)” is issued, and the axis synchronous control positional deviation error alarm signal SYNER<Fn403.0> is output.
The alarm DS0001 is issued to the master axis and slave axis.

When bit 4 (SYA) of parameter No. 8301 is set to 0, the positional deviation limit value of the master axis and slave axis is checked even if a servo-off occurs during axis synchronous control.

1.6.6.2 Methods of alarm recovery by synchronization error check

To recover from an alarm issued as a result of synchronization error check, the method is different by setting parameter of bit 5 (SCAx) of parameter No.8304.

- When the bit 5 (SCAx) of parameter No.8304 is set to 1

When the bit 5 (SCAx) of parameter No.8304 is set to 1 (synchronous operation is used at all times), a synchronization error can be decreased by using the correction mode. When the correction mode is used, synchronization error check can be temporarily disabled, and a movement can be made along the master axis or slave axis to correct a synchronization error.
In the correction mode, synchronization error compensation and error check are not performed, so that an alarm DS0003, "SYNCHRONIZE ADJUST MODE" is issued as a warning.

1. Select the correction mode, and select an axis along which a movement is to be made by manual feed of a master axis . Set bit 2 (ADJx) of parameter No. 8304 of the master axis or slave axis to 1 to select the correction mode. Thus, by manual feed of a master axis , a movement can be made along the axis with this parameter set to 1.
   When this parameter is set to 1, the alarm DS0003 is issued.
2. Reset the synchronization error excessive alarm.
   In this state, synchronization error compensation and error check are not performed. Be careful.
3. Select the manual mode (jog, incremental feed, or handle).
4. While checking the synchronization error amount, make a movement along the master axis or slave axis in the direction that reduces the error.
   If one master axis has multiple slave axes, an attempt to reduce the synchronization error of one slave axis by master axis movement may increase the synchronization error of another slave axis, thus a movement in any direction is disabled. In such a case, by setting bit 4 (MVBx) of parameter No. 8304 to 1, a movement can be made in the direction that increases the synchronization error.
5. When the synchronization error is reduced to within the allowable value for suppressing the alarm, reset the value of bit 2 (ADJx) of parameter No. 8304 to the original value to switch from the correction mode to the normal synchronization mode.
   Synchronization error compensation and synchronization error check are restarted.
6. Reset the correction mode alarm.

- When the bit 5 (SCAx) of parameter No.8304 is set to 0

When the bit 5 (SCAx) of parameter No.8304 is set to 0 (synchronous operation and normal operation are switched by using an input signal), a synchronization error can be decreased by using the normal operation. Use the procedure below for recovery from alarm SV0005.

1. Set signal SYNC/SYNCJ to "0" to select normal operation.
2. Set a value greater than the current value in the parameter No. 8314 for specifying a maximum allowable synchronization error, then reset the alarm.
3. Make a movement along the master axis or slave axis by using the manual handle so that the machine coordinates of the master axis and slave axis becomes almost the same.
4. Return the value of parameter No. 8314 for specifying a maximum allowable synchronization error to the original value.

1.6.7 Axis Synchronous Control Torque Difference Alarm

Explanation

If a movement made along the master axis differs from a movement made along the slave axis during axis synchronous control, the machine can be damaged. To prevent such damage, the torque command difference between the two axes is observed. If the difference exceeds certain limit, a servo alarm SV0420, " SYNC TORQUE EXCESS" can be issued.

[System configuration]

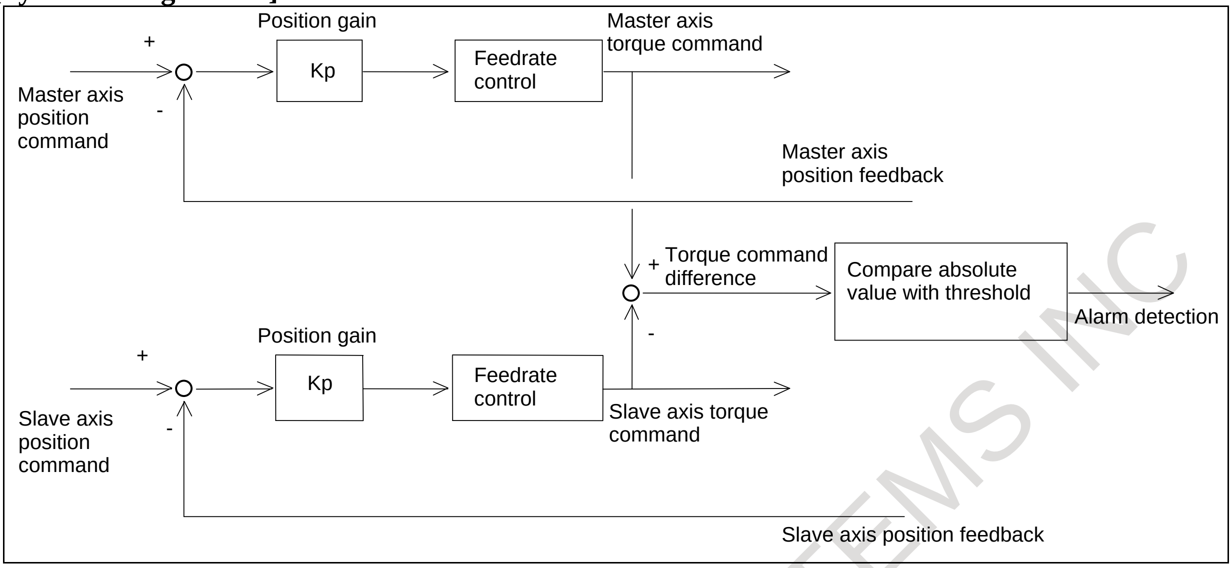


Fig. 1.6.7 (a) System configuration

[Method of use]

Specify the threshold parameter No. 2031 according to the procedure below.

1. Set 14564 in parameter No. 2031.
2. Display the diagnostic screen by pressing the function key then the soft key [DIAGNOSIS].
   Diagnosis data No. 3506 indicates the absolute value of the torque difference between the two axes.
3. Read the absolute torque difference value presented when normal operation is being performed. In the threshold parameter No. 2031, set a value obtained by adding some margin to the read absolute value.
   An absolute torque difference value can be read using SERVO GUIDE Mate.

- Enabling/disabling of alarm detection

Alarm detection is enabled when the time set in parameter No. 8327 has elapsed after the servo ready signal SA <Fn000.6> is set to "1". When the signal for disabling torque difference alarm detection for axis synchronous control NSYNCA <Gn059.7> is set to "1", alarm detection is disabled.

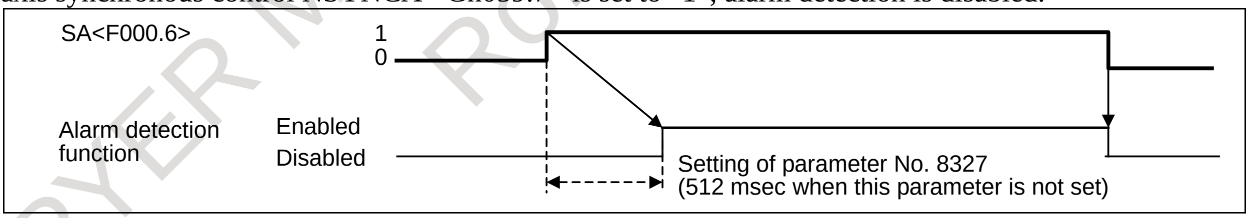


Fig.1 .6.7 (b) Timing chart

When the servo ready signal SA <Fn000.6>is set to "0", torque difference alarm detection is disabled.

> **NOTE**
> The servo axis number combination of the master axis and slave axis synchronized with each other must be such that an odd servo axis number is assigned to the master axis and the next servo axis number is assigned to the slave axis like (1,2) or (3,4).

Parameter

2031	Torque command difference threshold of torque difference alarm

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 14564

If the absolute value of the torque command difference between two axes exceeds the value set in this parameter, an alarm is issued.
Set the same value for two axes that are placed under axis synchronous control.
The servo axis numbers of the synchronized master axis and slave axis must be assigned so that an odd number is assigned to the master axis and the next axis number is assigned to the slave axis. Examples are (1,2) and (3,4).

Diagnosis data

3506	SYNC TORQUE DIFFERENCE

[Data type] Word axis
[Valid data range] 0 to 32767

The absolute torque difference value between the master axis and the slave axis in the axis synchronous control is displayed.

1.6.8 Synchronization Error Compensation

Explanation

When a synchronization error amount exceeding the zero width set in parameter No. 8333 is detected, compensation pulses for synchronization error reduction are calculated and added onto the command pulses output for the slave axis. This compensation is not performed in servo-off state, servo alarm state, follow-up operation, and correction mode.
Compensation pulses are calculated by multiplying the synchronization error amount between master and slave axes by a compensation gain.

Compensation pulses = synchronization error × (Ci/1024)
Ci: Compensation gain (parameter No. 8334)

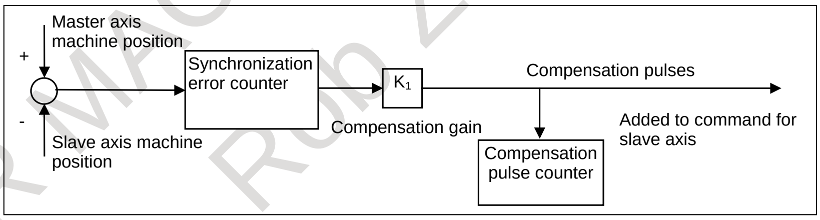


Fig. 1.6.8 (a) Compensation pulses

In case that the following parameters are set, after the reference position establishment had been completed and the first synchronization establishment after power-up was performed, the synchronization error counter is reset and the synchronization error compensation is started.

- Bit3 (CLPx) of parameter No.8304 is set to 1.
- Bit5 (SCAx) of parameter No.8304 is set to 1.
- The value other than 0 is set in parameter No.8334

NOTE
If the positional deviation corresponding to the load were generated like a past analog servo, this function was effective for decreasing the synchronization error. When a current digital servo is applied, positional deviation corresponding to the load are not generated. Therefore, the function need not be applied.

- Synchronization error compensation smooth suppress function

When bit 6 (SMSx) of parameter No. 8304 is set to 1, the synchronization error compensation smooth suppress function is enabled. With this function, another set of parameters for a synchronization error zero width and synchronization error compensation gain (B and Ks in the Fig.1.6.8 (b)) can be set. So, even a small synchronization error can be reduced smoothly as shown below.

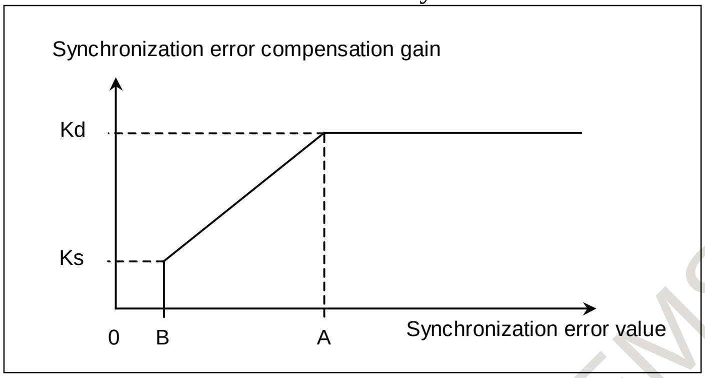


Fig. 1.6.8 (b) Smooth suppress function

A: Synchronization error zero width (parameter No. 8333)
B: Synchronization error zero width 2 (parameter No. 8335) (0 < B < A)
Kd: Synchronization error compensation gain (parameter No. 8334)
Ks: Synchronization error compensation gain 2 (parameter No. 8336) (0 < Ks < Kd)
Er: Synchronization error amount between the current master axis and slave axis
K: Current synchronization error compensation gain for Er

1. When Er < B, compensation is not performed. (K = 0)
2. When B < Er < A
   Compensation is performed with the following gain:
   $$K = Ks + \frac{(Er - B)(Kd - Ks)}{A - B}$$
3. When Er > A, compensation is performed with a gain of K = Kd.

- Synchronization establishment

When synchronization error compensation is performed , synchronization establishment is performed in the same way as synchronization error compensation. This means that the positional difference between the master axis and slave axis is regarded as a synchronization error, and pulses produced by multiplying the positional difference by a synchronization error compensation gain are output for the slave axis. So, the parameter No. 8334 for specifying a synchronization error compensation gain must be set before synchronization establishment can be performed.
If the parameter No. 8333 for specifying a synchronization error compensation zero width is set, no further synchronization establishment is performed after the positional difference between the master axis and slave axis becomes the zero width or below.
For “first synchronization establishment after power-up” and “synchronization establishment after emergency stop cancellation, etc.”, see Subsection “Synchronization Establishment”.

- Synchronization error check

When synchronization error compensation is performed, the synchronization error check is performed the check considering a positional deviation. The actual machine position shift considering a servo positional deviation as well is checked. Depending on the value of a synchronization error, one of two alarms is issued: an alarm (DS alarm) for deceleration stop, and an alarm (SV alarm) for turning off the servo system immediately. This check is enabled when a value other than 0 is set in the parameters Nos. 8331 and 8332 for specifying a maximum allowable synchronization error. When using this method of checking a synchronization error amount, disable "Synchronization error check based on machine coordinates" in Subsection by setting parameter No. 8314 to 0.

Synchronization error excessive alarm 1 (DS0002)

When a synchronization error exceeding the value set in parameter No. 8331 is detected, synchronization error excessive alarm 1 is issued. When synchronization error excessive alarm 1 is issued, the motor gradually stops. At this time, synchronization error compensation remains enabled, so that the synchronization error is reduced by compensation. Accordingly, as the synchronization error amount becomes smaller than the maximum allowable value, the alarm can be reset. If the alarm cannot be reset, the synchronization error needs to be manually corrected by selecting the correction mode in Subsection.

Synchronization error excessive alarm 2 (SV0002)

When a synchronization error exceeding the value set in parameter No. 8332 is detected, synchronization error excessive alarm 2 is issued. Before synchronization establishment is performed at power-up time, a value obtained by multiplying the value set in parameter No. 8332 by the coefficient set in parameter No. 8330 is used for judgment. When synchronization error excessive alarm 2 is issued, the motor immediately stops as in the case of other servo alarms. Accordingly, the positional difference between the master axis and slave axis remains uncorrected, so that the alarm cannot be reset usually. In this case, the synchronization error needs to be manually corrected by selecting the correction mode in Subsection.

1.6.9 Combination with other functions

Explanation

Notes on setting parameters for each axis

Parameters to be set for each axis can be divided into the following four types when they are set for an axis under axis synchronous control:

(1) Parameter which must be set the same value to the master and slave axes
(2) Parameter which needs to be set only to the master axis (data to the slave axis is not used.)
(3) Parameter which may be set different values to the master and slave axes
(4) Parameter which needs to be set only to the slave axis (data to the master axis is not used.)

Parameters are listed for each type below. If a parameter is not listed in any table below, assume that the parameter is of type (1) and set the same value to the master and slave axes.

> ⚠ **CAUTION**
> 1 If different values are set in a parameter of type (1) for the master and slave axes, these axes may not operate as axes under axis synchronous control.
> 2 When a signal is used to switch between synchronous and normal operation, in a parameter of type (2), also set a value for each of the master and slave axes.

(1) Parameters which must be set to the same value for the master and slave axes

Parameter number	Description
12#7	Releasing the assignment of the controlled axis for each axis
1005#0	Whether reference position return has been performed
1005#1	Enabling setting the reference position without dogs
1005#4	Enabling the external deceleration signal for the positive direction in cutting feed for each axis
1005#5	Enabling the external deceleration signal for the negative direction in cutting feed for each axis
1005#6	Turning off MCC signal of the servo amplifier in axis detach
1005#7	Enabling the controlled axis detach signal for each axis
1006#0,#1	Setting a rotary axis
1006#3	Specifying move commands for each axis using diameter programming
1006#5	Direction of manual reference position return
1007#1	Using the same method as for manual reference position return to perform automatic reference position return (G28)
1007#3	The rotary axis command in an absolute command is the absolute position of the specified value for the end point coordinates and the sign of the specified value for the direction of rotation.

Parameter number	Description
1007#4	Method of setting the reference position without dogs
1008#0	Enabling rotary axis roll-over
1008#1	Not making a short cut in an absolute command
1008#2	Rounding relative coordinates with the travel distance of one rotation.
1240	Coordinate value of the reference position in the machine coordinate system
1241	Coordinate value of the second reference position in the machine coordinate system
1242	Coordinate value of the third reference position in the machine coordinate system
1243	Coordinate value of the fourth reference position in the machine coordinate system
1260	Amount of a shift per one rotation of a rotary axis
1310#0,#1	Enabling stored stroke check 2, 3
1320	Coordinate value I of stored stroke check 1 in the positive direction on each axis
1321	Coordinate value I of stored stroke check 1 in the negative direction on each axis
1322	Coordinate value of stored stroke check 2 in the positive direction on each axis
1323	Coordinate value of stored stroke check 2 in the negative direction on each axis
1324	Coordinate value of stored stroke check 3 in the positive direction on each axis
1325	Coordinate value of stored stroke check 3 in the negative direction on each axis
1326	Coordinate value II of stored stroke check 1 in the positive direction on each axis
1327	Coordinate value II of stored stroke check 1 in the negative direction on each axis
1420	Rapid traverse rate for each axis
1424	Manual rapid traverse rate for each axis (bit 0 of (SJR) parameter No.8306 = 1)
1610#0,#1	Acceleration type of cutting feed or dry run during cutting feed
1610#4	Acceleration type of jog feed
1620	Time constant used for linear acceleration/deceleration for each axis
1621	Time constant T2 used for bell-shaped acceleration/deceleration in rapid traverse for each axis
1622	Time constant of acceleration/deceleration in cutting feed for each axis
1623	FL rate of acceleration/deceleration in cutting feed for each axis
1624	Time constant of acceleration/deceleration in jog feed for each axis.
1625	FL rate of exponential acceleration/deceleration in jog feed for each axis
1626	Time constant of acceleration/deceleration during thread cutting cycle
1627	FL rate of acceleration/deceleration during thread cutting cycle
1671	Maximum permissible acceleration of acceleration/deceleration before interpolation for linear rapid traverse for each axis or permissible reference acceleration of optimum torque acceleration/deceleration
1763	FL rate acceleration/deceleration after cutting feed interpolation for each axis in the acceleration/deceleration before interpolation mode
1769	Time constant of acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode
1815#2	Using a linear scale with reference marks
1818#0,#1,#3, 1819#2	Related to the linear scale with absolute addressing reference marks/linear scale with an absolute addressing origin
1819#0	Follow-up in the servo off state
1819#1	Reference position establishment state at the time of a servo alarm
1821	Reference counter capacity
1825	Servo loop gain
1881	Group number when an unexpected disturbance torque is detected
1882	Intervals of mark 2 of a linear scale with absolute address reference marks.
1885	Maximum allowable value for total travel during torque control
1886	Positional deviation when torque control is canceled
2028	Limit speed for enabling position gain switching
2031	Torque-command-difference threshold for a torque-difference alarm
2060	Torque limit
2068	Feed forward coefficient
2092	Advanced preview feed forward coefficient
2144	Position feed forward coefficient for cutting

Parameter number	Description
2178	Position gain for rapid traverse
2179	Reference counter (denominator)
3605#0	Using bidirectional pitch error compensation
3605#1	Using interpolation type pitch error compensation
3605#2	Using interpolated straightness compensation
3624	Interval between pitch error compensation positions for each axis
3625	Travel distance per revolution in pitch error compensation of rotary axis type
7310	Order of axes to move to with dry run after a program restart (in the case of not switching between synchronous operation and normal operation with signals)
8304#0	Setting the reference position for the one direction synchronization compensation function
11242	Time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse
11248	Time constant 1 of acceleration/deceleration after interpolation in Smart overlap for each axis
11249	Time constant 2 of acceleration/deceleration after interpolation in Smart overlap for each axis
11814	The second servo loop gain for each axis
13622	Time constant for acceleration/deceleration after interpolation when AI contour control is used (precision level 1)
13623	Time constant for acceleration/deceleration after interpolation when AI contour control is used (precision level 10)

(2) Parameters which need to be set only for the master axis

Parameter number	Description
1005#3	Operation to be performed during a manual reference position return when a reference position is established
1012#0	Method of setting the reference position without dogs
1250	Coordinates of the reference position when automatic coordinate system setting is performed
1408#0	The rotary axis feedrate control method is to convert the rotation speed on the circumference of a virtual circle.
1421	F0 rate of rapid traverse override for each axis
1423	Feedrate in manual continuous feed (jog feed) for each axis
1424	Manual rapid traverse rate for each axis (bit 0 of (SJR) parameter No.8306 = 1)
1425	FL rate of the reference position return for each axis
1427	External deceleration rate of rapid traverse for each axis
1428	Reference position return speed for each axis
1430	Maximum cutting feedrate for each axis
1432	Maximum cutting feedrate for each axis in the mode of acceleration/deceleration before interpolation
1465	Radius of the virtual circle in a rotary axis virtual circle velocity command
1660	Maximum permissible acceleration of acceleration/deceleration before interpolation for each axis
1735	Permissible acceleration for each axis in the deceleration function with the acceleration in circular interpolation
1737	Permissible acceleration for each axis in the deceleration function with the acceleration in AI contour control
1783	Permissible speed difference in speed determination with the speed difference at a corner
1788	Permissible acceleration change amount in speed determination with the acceleration change on each axis
1789	Permissible acceleration change amount in speed determination with the acceleration change on each axis (linear interpolation)
3455#0	When the decimal point is omitted, mm, inch, sec units (calculator-type decimal point input)
5022	Distance (L) from reference tool tip position to the reference measurement surface
5401#0	Enabling scaling
5421	Scaling magnification for each axis
5440	Positioning direction and overrun distance in single directional positioning
7181	First retract distance at reference point setting with mechanical stopper

Parameter number	Description
7182	Second retract distance at reference point setting with mechanical stopper
7183	First retract speed at reference point setting with mechanical stopper
7184	Second retract speed at reference point setting with mechanical stopper
7185	Retract speed at reference point setting with mechanical stopper (common to the first and second retractions)
7741	Retract amount
8410	Allowable feedrate difference used for feedrate determination, based on a corner feedrate difference
19541 to 19544	Optimum torque acceleration/deceleration (speed)
19545 to 19568	Optimum torque acceleration/deceleration (acceleration)
19667	Control point shift vector

(3) Parameters which may be set to different values for the master and slave axes

Parameter number	Description
1020	Program axis name for each axis
1023	Number of the servo axis for each axis
18XX 2XXX	Parameters related to servo (other than those listed in (1), (2), or (4))
3115#0	Not displaying the current position
3115#1	Not displaying the absolute and relative coordinates
3115#3	Not adding axis moving to actual cutting feedrate display
3131	Subscript of axis name of synchronous controlled axis and tandem control axis
3620	Number of the pitch error compensation position for the reference position for each axis
3621	Number of the pitch error compensation position at extremely negative position for each axis
3622	Number of the pitch error compensation position at extremely positive position for each axis
3623	Magnification for pitch error compensation for each axis
3626	Number of the both-direction pitch error compensation position at extremely negative position (for movement in the negative direction)
3627	Pitch error compensation at reference position when a movement to the reference position is made from the direction opposite to the direction of reference position return
5861 to 5864	Inclination compensation : Compensation point number for each axis
5871 to 5874	Inclination compensation : Compensation at compensation point number for each axis
7186	Torque limit value in reference point setting with mechanical stopper (for 0% to 39%)
7187	Torque limit value in reference point setting with mechanical stopper (for 0 to 100%)
7310	Order of axes to move to with dry run after a program restart (in the case of switching between synchronous operation and normal operation with signals (SYNC1 to SYNC8<Gn138> and SYNCJ1 to SYNCJ8<Gn140>))
24096	Connector number for the first separate detector interface unit
24097	Connector number for the second separate detector interface unit
24098	Connector number for the third separate detector interface unit
24099	Connector number for the fourth separate detector interface unit
24100	Connector number for the fifth separate detector interface unit
24101	Connector number for the sixth separate detector interface unit
24102	Connector number for the seventh separate detector interface unit
24103	Connector number for the eighth separate detector interface unit

(4) Parameters which need to be set only for the slave axis

Parameter number	Description
1817#6	Performing tandem control for the axis
8303#0	Enabling automatic setting of grid positioning
8303#1	Setting the start of automatic setting of grid positioning
8303#2	Adding data for the slave axis for actual cutting feedrate display

Parameter number	Description
8303#7	Using the synchronization compensation function based on machine coordinates for the axis synchronous control function
8304#3	Enabling synchronization error compensation
8304#5	Always performing axis synchronous control
8304#6	Enabling the synchronization error smooth suppress function
8304#7	Enabling external machine coordinate system shift
8311	Axis number of master axis in axis synchronous control
8312	Enabling/disabling mirror image in axis synchronous control
8314	Maximum allowable error in synchronization error check based on machine coordinates
8323	Limit in positional deviation check in axis synchronous control
8325	Maximum compensation value in synchronization establishment based on machine coordinates
8326	Difference between master axis and slave axis reference counters
8327	Torque difference alarm detection timer
8331	Maximum allowable synchronization error for synchronization error excessive alarm 1
8332	Maximum allowable synchronization error for synchronization error excessive alarm 2
8333	Synchronization error zero width for each axis
8334	Synchronization error compensation gain for each axis
8335	Synchronization error zero width 2 for each axis
8336	Synchronization error compensation gain 2 for each axis

- How signals for each axis are handled

Some signals for each axis are input or output to both the master and slave axes and others are input or output to only the master axis. The following table lists the type of each signal.

- Input signals

Address	Bit	Symbol	Signal name	Master and slave axes	Master axis only
Gn100	0	+Jx	Feed axis direction select signals		O
Gn102	0	-Jx	Feed axis direction select signals		O
Gn104	0	+EXLx	Stored stroke limit 1 switching signals in axis direction	O	
Gn105	0	-EXLx	Stored stroke limit 1 switching signals in axis direction	O	
Gn106	0	MIx	Mirror image signals		O
Gn108	0	MLKx	Machine lock signal in each axis		O
Gn110	0	+LMx	Stroke limit external setting signals	O	
Gn112	0	-LMx	Stroke limit external setting signals	O	
Gn114	0	*+Lx	Over travel signals		O
Gn116	0	*-Lx	Over travel signals		O
Gn118	0	*+Edx	External deceleration signals		O
Gn120	0	*-Edx	External deceleration signals		O
Gn124	0	DTCHx	Controlled axes detach signals	O *1	
Gn126	0	*SVFx	Servo-off signals	O *1	
Gn130	0	*ITx	Interlock signals in each axis		O
Gn138	0	SYNCx	Signals for selecting the axis for axis synchronous control	O	
Gn140	0	SYNCJx	Signals for selecting the manual feed axis for axis synchronous control	O	
Gn192	0	IGVRYx	VRDY off alarm ignore signal in each axis	O *1	
X009	0	*DECx	Deceleration signals for reference position return		O

- Output signals

Address	Bit	Symbol	Signal name	Master and slave axes	Master axis only
Fn094	0	ZPx	Reference position return completion signals	O	
Fn096	0	ZP2x	2nd reference position return completion signals	O	

Address	Bit	Symbol	Signal name	Master and slave axes	Master axis only
Fn098	0	ZP3x	3rd reference position return completion signals	O	
Fn100	0	ZP4x	4th reference position return completion signals	O	
Fn102	0	MVx	Axis moving signals	O	
Fn104	0	INPx	In-position signals	O	
Fn106	0	MVDx	Axis moving direction signals	O	
Fn108	0	MMIx	Mirror image check signals	O	
Fn110	0	MDTCHx	Controlled axis detach status signals	O	
Fn120	0	ZRFx	Reference position establishment signals	O	

NOTE

1. Turn the signal marked with *1 on or off simultaneously for the master and slave axes in the synchronization mode.
2. In the above table, the address only for the 1st axis is listed. For the addresses of the 2nd and subsequent axes, see Appendix, "INTERFACE BETWEEN CNC AND PMC" in this manual.

- Axis selection in actual cutting feedrate display

The slave axis of the axis synchronous control does not execute the calculation of the actual cutting feedrate display during the synchronous operation.
By setting bit 2 (SAFx) of parameter No. 8303 to 1 for a slave axis, the slave axis can be included in an actual cutting feedrate display calculation during synchronous operation.

- Program restart

If an operation mode is not switched between synchronous operation and normal operation with signals (SYNC1 to SYNC8<Gn138> and SYNCJ1 to SYNCJ8<Gn140>) (to be in the synchronous state at all times), set the same data for the master axis and slave axis in parameter No.7310.
If an operation mode is switched between synchronous operation and normal operation with signals, the same data need not be set for the master axis and slave axis, but data (1 to the number of controlled axes) must be set for both the master axis and slave axis.

- Flexible synchronization/Superimposed control

The slave axis of flexible synchronization control or the slave axis of superimposed control can be set to the master axis of axis synchronous control by setting bit 0 (FSS) of parameter No. 8307 to 1. Then, it is necessary to execute beginning and the end of flexible synchronization control or superimposed control while axis synchronous control. The example of machining program is shown as follows.

Example of the machining program)

```
:
:
M50;      : Beginning of axis synchronous control
M210;     : Turning Superimposed ahead signals on (In case of superimposed control)
M16;      : Beginning of flexible synchronization control or superimposed control
:
:
M17;      : End of flexible synchronization control or superimposed control
M211;     : Turning Superimposed ahead signals off (In case of superimposed control)
M51;      : End of axis synchronous control
```

- Reference position return of axis synchronous control

It is necessary to perform reference position return before flexible synchronization control or superimposed control is begun.

- Synchronization Establishment of axis synchronous control

It is necessary to perform synchronization Establishment before flexible synchronization control or superimposed control is begun.

- When using slave axis of superimposed control as master axis of axis synchronous control

It is necessary to set superimposed ahead signals OVLN <Gn531.4> to "1" during superimposed control is begun.

Note

1) Please turn on the mode of axis synchronous control before turning on the mode of flexible synchronization control or superimposed control when the slave axis of flexible synchronization control or the slave axis of superimposed control is used as the master axis of axis synchronous control.
2) It is not possible to instruct the workpiece coordinate system setting command (G92) and local coordinate system setting command (G52) while the master axis of axis synchronous control is mode of flexible synchronization control or superimposed control.
3) When bit 0 (FSS) of parameter No. 8307 is set to 1, the display of slave axis of axis synchronous control is updated with the amount of travel after acceleration/deceleration and, therefore, the display may appear not to be in synchronization.
4) When the slave axis of flexible synchronization control or the slave axis of superimposed control is used as the master axis of axis synchronous control, slave axis of axis synchronous control checks the state of stopping state by positioning deviation limit in movement.
5) The output of the in-position signals INP1 to INP8 <Fn104> and the axis moving signals MV1 to MV8 <Fn102> follows each specification of flexible synchronization control or superimposed control.
6) At the correction mode inside (bit 2 (ADJ) of parameter No. 8304 = 1), the movement instruction cannot be done to the master axis of flexible synchronization control or the master axis of superimposed control.
7) Use an M code preventing buffering as the M code for turning the superimposed ahead signals on or off.

limitation

When bit 0 (FSS) of parameter No. 8307 is set to 1, the following parameters become invalid for the master axis and the slave axis of axis synchronous control.

Parameter number	Description
12255	Maximum servo motor speed
12256	Maximum servo motor acceleration rate

When bit 0 (FSS) of parameter No. 8307 is set to 1, please set the same value to master axis and slave axis of axis synchronous control about the following parameters.

Parameter number	Description
1013	Parameter of increment system
18xx	Parameter related to servo
2xxx	

Flexible synchronization control / superimposed control

When this function is effective(bit 0 (FSS) of parameter No. 8307 = 1), slave axis of axis synchronous control cannot be made the axis of flexible synchronization control or the axis of superimposed control.

General purpose retract

If general purpose retract is used in flexible synchronization control, general purpose retract can be set only to the master axis. General purpose retract can not be used in superimposed control. Because of this, general purpose retract cannot be used in the master axis of axis synchronous control while using this function.

Tandem control

When this function is effective(bit 0 (FSS) of parameter No. 8307 = 1), tandem control cannot be used with a related axis of axis synchronous control.

1.6.10 Automatic Slave Axis Parameter Setting

Explanation

Axis synchronous control involves parameters which must be set to the same values for the master axis and slave axis. (See the parameter list previously described in "Parameters which must be set to the same value for the master and slave axes"). If bit 4 (SYPx) of parameter No. 8303 is 1, setting values for these parameters for the master axis causes the same values to be set to the parameters for the slave axis.
This function is enabled when parameters are set with the following methods:

- MDI key input
- Parameter file input
- Input with a program command
  (1) Programmable data input (parameter)
  (2) Stored stroke check 2 on (parameters Nos. 1322, 1323)
  (3) Cutting condition selection function (parameters Nos. 1769, 13622, and 13623)
- Input with a signal
  Stroke limit external setting (parameters Nos. 1320, 1321)
- Input with the FOCAS function and the window function

- Prohibition of writing for the slave axis

If bit 4 (SYPx) of parameter No. 8303 is 1, it is prohibited to set values to the parameters for the slave axis previously described in "Parameters which must be set to the same value for the master and slave axes". Even if values are set, they will be invalid.
If any parameter other than the BIT parameter is set with the methods below, either an alarm or a warning will be output.
MDI key input
A "WRITE PROTECT" warning is issued.
Programmable data input, stored stroke check 2 on
The alarm PS5379, " WRITE PROTECTED TO SLAVE AXIS" is issued.
Input with the FOCAS function and the window function
The return value is 7 (write protect error).

1.6.11 Torque Difference of Axis Synchronous

Explanation

By acquiring and comparing information with this system variable at multiple points, can be detected balanced synchronous position in master axis and slave axis.
This function can read information of toque difference of master axis and slave axis in Axis synchronous control with system variable (#180801 to #180850).

Information of torque difference of master axis and slave axis

System variable number	System variable name	Attribute	Description
#180801 to #180850	[#_SYNTD[n]]	R	Torque difference of master axis and slave axis in Axis synchronous control Range：-32768 to 32767 Unit：Unit in which the set value corresponding to Nominal current limit of Amplifier [Ap] is 6554. (Note) Subscript n represents a compensation number (1 to 50).

R is attribute of a variable and indicate read-only.

Parameter (No.8311) determines the combination of master axis and slave axis, and torque difference of master axis and slave axis is output to system variable of slave axis during synchronous operation and normal operation.
The output torque difference is the value obtained by subtracting torque value of slave axis from the torque value of master axis.
System value is read “0” in master axis. The case of axis more than the number of axis in a path is read “0” as with master axis.
When value of parameter (No.8311) is incorrect, data of system variable cannot be read and becomes alarm (PS5580) “CANNOT GET SYSTEM VARIABLE”.

Application example

When setting Pitch error compensation of Axis synchronous control using this function. Acquiring torque difference master axis and slave axis from system variable, detecting the position where torque difference of master axis and slave axis is the minimum, it is possible to set more optimal Pitch error compensation.

Example)

Setting procedure of Pitch error compensation value in slave axis with this system variable (#180801 to #180850) is as follows.

In using this function, enable synchronization establishment of Axis synchronous control in NC parameter.

1 Set synchronization establishment of maser axis using Laser measuring instrument.
2 In synchronous operation, move master axis to the first observation point.
3 After cancel synchronous operation, move slave axis only while acquiring torque difference by system variable (#180801 to #180850), find position where torque difference of master axis and slave axis is the smallest
4 After save coordinate difference of master axis and slave axis to work variable, set synchronous operation.
5 Operate “3” to “4” in all master axis of observation point.
6 Calculate pitch data amount of movement of slave axis and set to Pitch error compensation of slave axis.

Limitation

- Real time custom macro

System variables (#180801 to #180850) can’t use in real time custom macro. Value is read “0” in Real time custom macro.

- Flexible path axis assignment

System variable (#180801 to #180850) corresponds to Flexible path axis assignment if axis configuration is changed while maintaining parameter (No.8311) and axis configuration.
Correspondence example of Flexible path axis assignment is as follows.

Basic configuration

Axis name	Axis number	Parameter(No.8311)	Description
Xm	1	0	Master axis of Xs-axis
Xs	2	1	Slave axis of Xm-axis

(Example1 : Add Y-axis after Xm-axis to basic configuration.)

When setting value of Parameter No.8311) and axis configuration are in the same states by Flexible path axis assignment, system variable is read torque difference of master axis and slave axis.

Xm	1	0	Master axis of Xs-axis
Y	- →2	0	Add axis
Xs	2→3	1	Slave axis of Xm-axis (Axis number and Parameter(No.8311) is match)

(Example2 : Add Y-axis before Xm-axis to basic configuration.)

When setting value of Parameter No.8311) and axis configuration are in different states by Flexible path axis assignment, system variable is read “0”.

Y	- →1	0	Add axis
Xm	1→2	0	Master axis of Xs-axis (Axis number of master axis changes)
Xs	2→3	1	Slave axis of Xm-axis (Axis number and Parameter(No.8311) don’t match)

in the case where axis of synchronization relationship moves to another path, system variable is read “0”.

- Background operation

System variables (#180801 to #180850) can’t use in Background operation. Value is “0” in Background operation.

1.6.12 Signal

Signals for selecting the axis for axis synchronous control
SYNC1 to SYNC8<Gn138>

[Classification] Input signal

[Function] Axis synchronous control is performed during memory or DNC or MDI operation.
This signal is provided for each controlled axis. The number at the end of the signal name represents the number of the controlled axis.

SYNC<u>x</u>

x : 1 .....The first axis becomes the slave axis for axis synchronous control.
2 .....The second axis becomes the slave axis for axis synchronous control.
3 .....The third axis becomes the slave axis for axis synchronous control.
: :

[Operation] When this signal is set to “1”, the control unit operates as described below:

- During memory or DNC or MDI operation, the control unit issues the move command specified for the master axis to both the master axis and slave axis of axis synchronous control.

The master axis is specified with a parameter.

See the following timing charts about the sequence that changes the synchronous operation and the normal operation.

Signals for selecting the manual feed axis for axis synchronous control SYNCJ1 to SYNCJ8<Gn140>

[Classification] Input signal

[Function] Axis synchronous control is performed in jog, handle, or incremental feed mode, or manual reference position return. This signal is provided for each controlled axis. The number at the end of the signal name represents the number of the controlled axis.

SYNCJx

x : 1 .....The first axis becomes the slave axis for axis synchronous control.
2 .....The second axis becomes the slave axis for axis synchronous control.
3 .....The third axis becomes the slave axis for axis synchronous control.
: :

[Operation] When this signal is set to “1”, the control unit operates as described below:

- In jog, handle, or incremental feed mode, the control unit issues the move command specified for the master axis to both the master axis and slave axis of axis synchronous control.

The master axis is specified with a parameter.

See the following timing charts about the sequence that changes the synchronous operation and the normal operation.

Signal for disabling torque difference alarm detection for axis synchronous control NSYNCA<Gn059.7>

[Classification] Input signal

[Function] When the torque difference alarm function for axis synchronous control is used, this signal can be used to disable alarm detection.

[Operation] When this signal is set to “1”, torque difference alarm detection for axis synchronous control is disabled.

Machine coordinate match state output signals SYNMT1 to SYNMT8<Fn210>

[Classification] Output signal

[Function] When master/slave axis pairs are set for axis synchronous control, this signal notifies an external unit that the machine coordinates of the master axis is adjusted to those of the slave axis for each pair, regardless of the synchronous operation on or off state and servo ready state.

[Operation] When this signal is set to “1”, the machine coordinates of the master axis is adjusted to those of the slave axis. The signal corresponding to the pair of a master axis and the slave axis with the lowest axis number is output first and the machine coordinate status of up to eight pairs can be checked.

Synchronization compensation enable state output signals SYNOF1 to SYNOF8<Fn211>

[Classification] Output signal

[Function] When master/slave axis pairs are set for axis synchronous control, this signal notifies an external unit that the machine coordinates difference between the master and slave axes is less than or equal to the maximum compensation for synchronization for each pair, regardless of the synchronous operation on or off state.

[Operation] When this signal is set to “1”, the machine coordinates difference between the master and slave axes is less than or equal to the maximum compensation for synchronization. When the servo ready signal SA<Fn000.6> is set to "0", this signal is set to "0". The signal corresponding to the pair of a master axis and the slave axis with the lowest axis number is output first and whether to enable synchronization compensation can be checked for up to eight pairs. This signal is not output for each axis.

Signal for indicating a positional deviation error alarm for axis synchronous control SYNER<Fn403.0>

[Classification] Output signal

[Function] When the positional deviation check function is used for axis synchronous control, this signal notifies an external unit that the alarm is issued.

[Operation] When axis synchronous control is applied, the servo positional deviation of the master axis and that of the slave axis are monitored. If the limit set in parameter No. 8323 is exceeded, alarm DS0001 is issued and the signal for indicating a positional deviation error alarm for axis synchronous control <Fn403.0> is set to “1”.
This signal is set to “0” when the alarm is cleared by a reset. This signal is not output for each signal.

Axis synchronous control status signals SYNO1 to SYNO8<Fn532>

[Classification] Output signal

[Function] These signals notify that axis synchronous control is in progress.

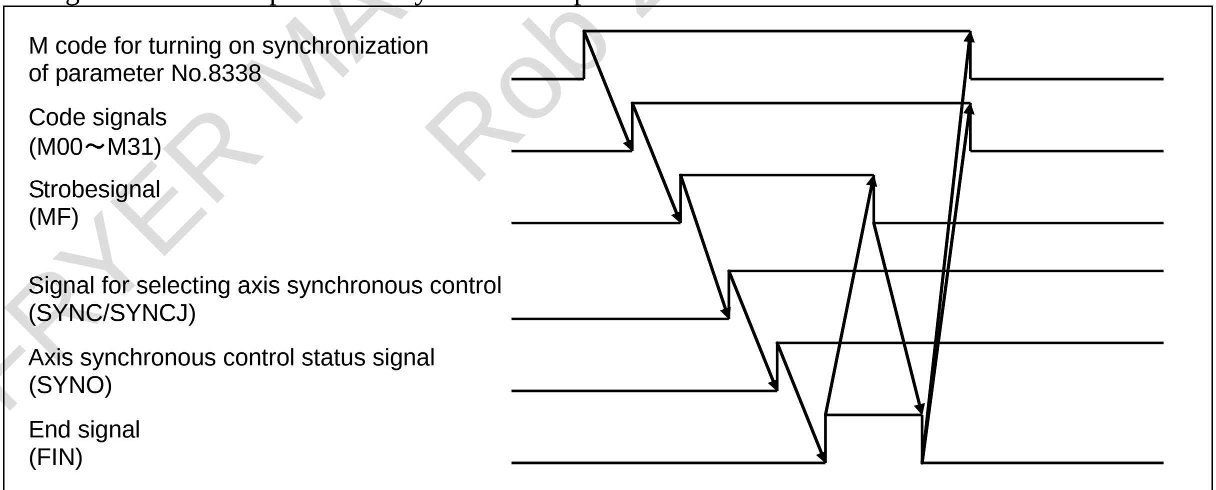
[Operation] These signals become “1” in the following case:
- When the corresponding axis is in axis synchronous control.

They become “0” in the following case:
- When the corresponding axis is not in the axis synchronous control.

> **NOTE**
> Whether axis synchronous control is in progress does not necessarily correspond to individual selection signals/parameters (axis synchronous control selection signal, axis synchronous control manual feed selection signal, bit 5 (SCAx) of parameter No. 8304). This signal is 0 in the emergency stop, servo alarm, serve off, and axis detach states.

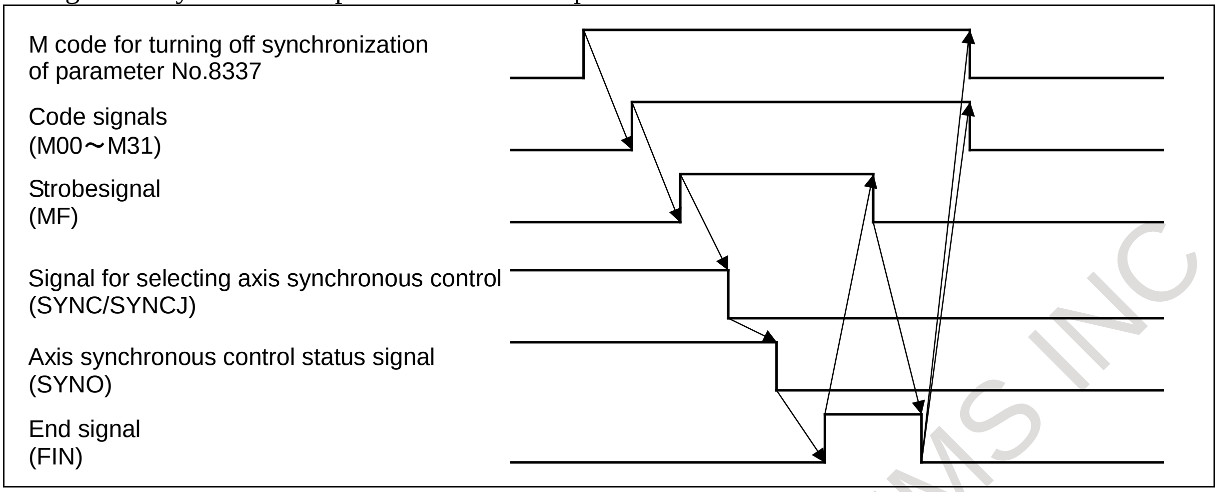
- Timing charts

Change from normal operation to synchronous operation



Supplement : Synchronization compensation enable state output signal SYNOF and machine coordinate match state output signals SYNMT can be check before signal SYNC/SYNCJ is set to “1”.

Change from synchronous operation to normal operation



Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn059	NSYNCA							
Gn138	SYNC8	SYNC7	SYNC6	SYNC5	SYNC4	SYNC3	SYNC2	SYNC1
Gn140	SYNCJ8	SYNCJ7	SYNCJ6	SYNCJ5	SYNCJ4	SYNCJ3	SYNCJ2	SYNCJ1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn210	SYNMT8	SYNMT7	SYNMT6	SYNMT5	SYNMT4	SYNMT3	SYNMT2	SYNMT1
Fn211	SYNOF8	SYNOF7	SYNOF6	SYNOF5	SYNOF4	SYNOF3	SYNOF2	SYNOF1
Fn403								SYNER
Fn532	SYNO8	SYNO7	SYNO6	SYNO5	SYNO4	SYNO3	SYNO2	SYNO1

1.6.13 Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1000								EEA

[Input type] Parameter input
[Data type] Bit

#0 EEA An extended axis name and extended spindle name are:
0: Invalid
1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
1008				SFDx				

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#4 SFDx In reference position return based on the grid method, the reference position shift function is:
0: Disabled
1: Enabled

1025	Program axis name 2 for each axis
1026	Program axis name 3 for each axis

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 48 to 57, 65 to 90
When axis name extension is enabled (when bit 0 (EEA) of parameter No. 1000 is set to 1), the length of an axis name can be extended to a maximum of three characters by setting axis name 2 and axis name 3.

NOTE
If program axis name 2 is not set, program axis name 3 is invalid.

	#7	#6	#5	#4	#3	#2	#1	#0
1401							LRP	

[Input type] Parameter input
[Data type] Bit path

#1 LRP Positioning (G00)
0: Positioning is performed with non-linear type positioning so that the tool moves along each axis independently at rapid traverse.
1: Positioning is performed with linear interpolation so that the tool moves in a straight line.
When using 3-dimensional coordinate system conversion, set this parameter to 1.

1420	Rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1424	Manual rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rate of manual rapid traverse when the rapid traverse override is 100% for each axis.

NOTE
1. If 0 is set, the rate set in parameter No. 1420 (rapid traverse rate for each axis) is assumed.
2. When manual rapid traverse is selected (bit 0 (RPD) of parameter No. 1401 is set to 1), manual feed is performed at the feedrate set in this parameter, regardless of the setting of bit 4 (JRV) of parameter No. 1402.

	#7	#6	#5	#4	#3	#2	#1	#0
1603				**PRT**				

[Input type] Parameter input
[Data type] Bit path

#4 PRT For positioning of linear interpolation type:
0: Acceleration/deceleration of acceleration fixed type is used.
1: Acceleration/deceleration of time fixed type is used.

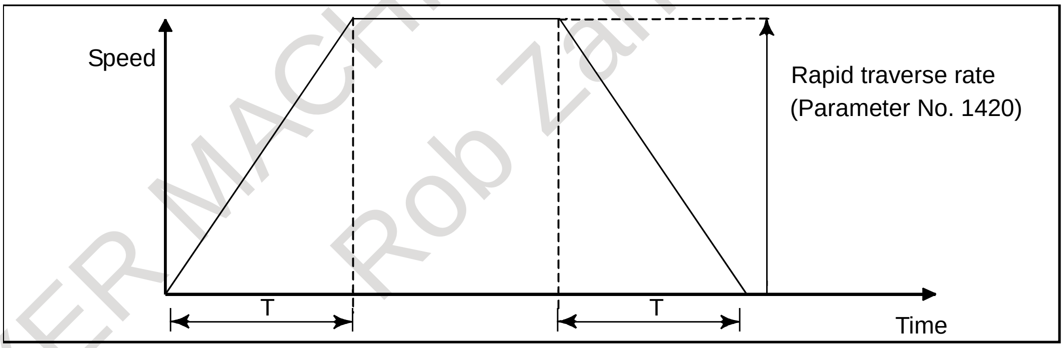
1620	**Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Specify a time constant used for acceleration/deceleration in rapid traverse.

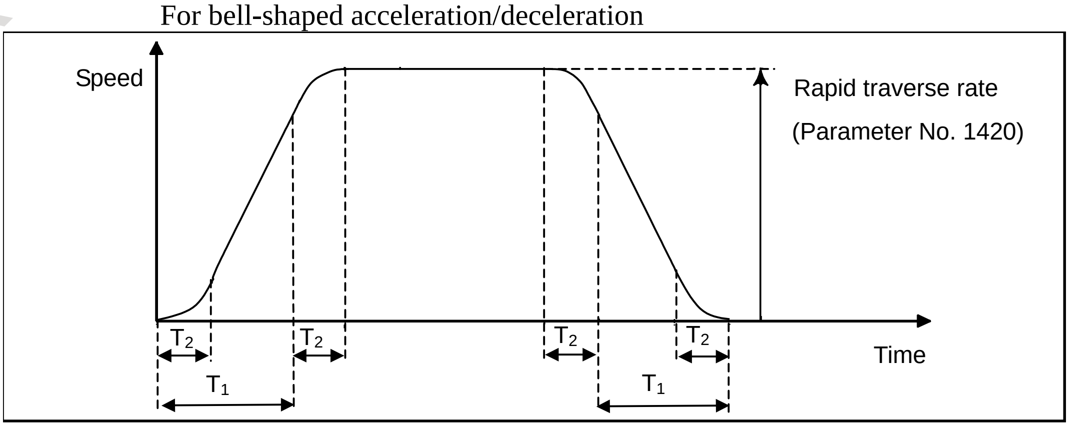
[Example]

For linear acceleration/deceleration



T : Setting of parameter No. 1620

For bell-shaped acceleration/deceleration



T_1 : Setting of parameter No. 1620
T_2 : Setting of parameter No. 1621

(However, $T_1 \geq T_2$ must be satisfied.)

Total acceleration (deceleration) time : $T_1 + T_2$
Time for linear portion : $T_1 - T_2$
Time for curve portion : $T_2 \times 2$

1621	Time constant T_2 used for bell-shaped acceleration/deceleration in rapid traverse for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 512

Specify time constant T_2 used for bell-shaped acceleration/ deceleration in rapid traverse for each axis.

1844	Distance to the first grid point when the reference position shift amount in the reference position shift function is 0 or when a reference position return is made by grid shift

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999999999 to 999999999

(1) When the reference position shift function is enabled (when bit 4 (SFDx) of parameter No. 1008 is set to 1)
Set the distance (detection unit) to the first grid point from a point at which the deceleration dog is released when the reference position shift (parameter No. 1850) is set to 0.

(2) When a reference position return is made by grid shift with a setting not to use reference position setting without dogs (when bit 4 (SFDx) of parameter No. 1008 is set to 0, and bit 1 (DLZx) of parameter No. 1005 is set to 0)
Set the distance to the first grid point from a point at which the deceleration dog is released. (Detection unit)

(3) When a reference position return is made by grid shift with a setting to use reference position setting without dogs (when bit 4 (SFDx) of parameter No. 1008 is set to 0, and bit 1 (DLZx) of parameter No. 1005 is set to 1)
Set the distance from the start position for reference position setting without dogs to the first grid point. (Detection unit)

NOTE
1 When the reference position shift function is enabled (when bit 4 (SFDx) of parameter No. 1008 is set to 1)
When bit 4 (SFDx) of parameter No. 1008 is set to 1, the distance from a point at which the deceleration dog is released to the first grid point (parameter No. 1844) is set to 0, and reference position shift (parameter No. 1850) is set to 0, a manual reference position return allows this parameter to be set automatically. Do not change an automatically set value.

NOTE

2 When a reference position return is made by grid shift with a setting not to use reference position setting without dogs (when bit 4 (SFDx) of parameter No. 1008 is set to 0, and bit 1 (DLZx) of parameter No. 1005 is set to 0)
When a manual reference position return using deceleration dogs is made, this parameter is set automatically.

3 When a reference position return is made by grid shift with a setting to use reference position setting without dogs (when bit 4 (SFDx) of parameter No. 1008 is set to 0, and bit 1 (DLZx) of parameter No. 1005 is set to 1)
When a reference position setting without dogs is made, this parameter is set automatically.

1850	Grid shift and reference position shift for each axis

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -99999999 to 99999999

To shift the reference position, the grid can be shifted by the amount set in this parameter. Up to the maximum value counted by the reference counter can be specified as the grid shift.
In case of bit 4 (SFDx) of parameter No. 1008 is 0: Grid shift
In case of bit 4 (SFDx) of parameter No. 1008 is 1: Reference point shift

NOTE

For setting the reference position without dogs, only the grid shift function can be used.
(The reference position shift function cannot be used.)

2031	Torque command difference threshold of torque difference alarm

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 14564

If the absolute value of the torque command difference between two axes exceeds the value set in this parameter, an alarm is issued.
Set the same value for two axes that are placed under axis synchronous control.
The servo axis numbers of the synchronized master axis and slave axis must be assigned so that an odd number is assigned to the master axis and the next axis number is assigned to the slave axis. Examples are (1,2) and (3,4).

3115							NDAx	NDPx

[Input type] Parameter input
[Data type] Bit axis

#0 NDPx The current position is:
0: Displayed.
1: Not displayed.

#1 NDAx The current position and the amount of the movement to be made in absolute and relative coordinates are:
0: Displayed.
1: Not displayed.

3130	Axis display order for current position display screens

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 32

Set the order in which axes are displayed on current position display screens (absolute, relative overall, and handle interrupt screens).

3131	Subscript of axis name

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 9, 65 to 90

In order to distinguish axes under synchronization control and tandem control, specify a subscript for each axis name.

Setting value	Meaning
0	Each axis is set as an axis other than a synchronization control axis and tandem control axis.
1 to 9	A set value is used as a subscript.
65 to 90	A set letter (ASCII code) is used as a subscript.

[Example] When the axis name is X, a subscript is added as indicated below.

Setting value	Axis name displayed on a screen such as the position display screen
0	X
1	X1
77	XM
83	XS

If a multi-path system is used, no extended axis name is used within a path, and no subscript is set for the axis names, then the path number is automatically used as the subscript for the axis names. To disable the display of axis name subscripts, set a blank (32) of ASCII code in the parameter for specifying an axis name subscript.

> **NOTE**
> If even one axis in a path uses an extended axis name when bit 2 (EAS) of parameter No. 11308 is set to 0, subscripts cannot be used for axis names in the path.

	#7	#6	#5	#4	#3	#2	#1	#0
8301				SYA				

[Input type] Parameter input
[Data type] Bit path

#4 SYA In the servo-off state in axis synchronous control, the limit of the difference between the positioning deviation of the master axis and that of the slave axis is:
0: Checked.
1: Not checked.

	#7	#6	#5	#4	#3	#2	#1	#0
8302	SMA							

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#7 SMA When an absolute position detector is attached, and bit 4 (APZx) of parameter No. 1815 for an axis in synchronous operation is set to 0, APZx of the pairing axis in synchronous operation is:
0: Not set to 0.
1: Set to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
8303	SOFx			SYPx		SAFx	ATSx	ATEx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ATEx In axis synchronous control, automatic setting for grid positioning is:
0: Disabled
1: Enabled
The setting for the slave axis is available.

#1 ATSx In axis synchronous control, automatic setting for grid positioning is:
0: Not started
1: Started
The setting for the slave axis is available.

> **NOTE**
> When starting automatic setting for grid positioning, set ATS to 1. Upon the completion of setting, ATS is automatically set to 0.

#2 SAFx In axis synchronous control, a movement along a slave axis is:
0: Not added to actual feedrate display.
1: Added to actual feedrate display.
The setting for the slave axis is available.

#4 SYPx In axis synchronous control, some parameters must be set to the same value for the master and slave axes. When a value is set in such a parameter for the master axis:
0: The same value is not automatically set in the parameter for the slave axis.
1: The same value is automatically set in the parameter for the slave axis.

> **NOTE**
> 1 The parameters that are automatically set are found previously described in "Parameters which must be set to the same value for the master and slave axes".
> 2 Set this parameter to the same value for both the master and slave axes.

#7 SOFx In axis synchronous control, the synchronization establishment function based on machine coordinates is:
0: Disabled.
1: Enabled.
The setting for the slave axis is available.When using synchronization error compensation, set this parameter to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
8304	SYEx	SMSx	SCAx	MVBx	CLPx	ADJx		SSAx

[Input type] Parameter input
[Data type] Bit axis

#0 SSAx When the one-direction synchronization establishment function under axis synchronous control is used:
0: The axis with a larger machine coordinate is used as the reference.
1: The axis with a smaller machine coordinate is used as the reference.

> **NOTE**
> 1 When this parameter is set, the power must be turned off before operation is continued.
> 2 Set this parameter to the same value for both the master and slave axes.

#2 ADJx In axis synchronous control, this parameter specifies an axis along which a movement is made in the correction mode.
0: A movement is not made in the correction mode along the axis.
1: A movement is made in the correction mode along the axis.
When this parameter is set to 1, the correction mode is set.
Along an axis with this parameter set to 1, a movement is made by a move command for the master axis.
Set this parameter for one of the master and slave axes.
When there are multiple slave axes for one master axis, set this parameter to 1 for an axis with which a synchronization error excessive alarm is issued for recovery. If an alarm is issued with multiple axes, modify this parameter after recovery of one axis to recover another axis.

#3 CLPx In axis feed synchronous control, synchronization error compensation is:
0: Disabled.
1: Enabled.
The setting for the slave axis is available.

#4 MVBx In the correction mode, a move command in a direction that increases a synchronization error is:
0: Ignored.
1: Valid.
When there are multiple slave axes for one master axis, an attempt to reduce the synchronous error of a slave axis by a movement along the master axis can increase the synchronization error of another slave axis. If this parameter is set to 0 in such a case, a movement can be made in neither direction along the master axis. In this case, set bit 2 (ADJx) of parameter No. 8304 to make a movement along a slave axis to perform a corrective operation.

#5 SCAx In axis synchronous control:
0: Synchronous operation is performed when the axis synchronous control manual feed selection signal SYNCJ <Gn140> or the axis synchronous control selection signal SYNC <Gn138> for slave axes is set to 1.
1: Synchronous operation is performed at all times.
The setting for the slave axis is available.

#6 SMSx The synchronization error smooth suppress function is:
0: Disabled.
1: Enabled.
The setting for the slave axis is available.

#7 SYEx When external machine coordinate system shift is specified by external data input/output for the master axis in synchronous control, the slave axis is:
0: Not shifted.
1: Shifted by the same amount as specified for the master axis.
The setting for the slave axis is available.
This function is disabled during normal operation.

	#7	#6	#5	#4	#3	#2	#1	#0
8305				SLR		SRF	SSE	SSO

[Input type] Parameter input
[Data type] Bit path

#0 SSO One-direction synchronization establishment function under axis synchronous control in axis synchronous control is:
0: Disabled.
1: Enabled.

#1 SSE After emergency stop, one-direction synchronization establishment function under axis synchronous control in axis synchronous control is:
0: Enabled.
1: Disabled.

#2 SRF In axis synchronous control, G28, G30, and G53:
0: Make the same movement along the slave axis as a movement along the master axis.
1: Make movements along the slave axis and master axis independently to specified positions.

#4 SLR When G28 is specified for an axis under axis synchronous control for which the reference position is not established:
0: Alarm PS0213, “ILLEGAL COMMAND IN SYNCHRO-MODE” is issued.

1: Reference position return operation of low-speed type is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
8306								SJR

[Input type] Parameter input
[Data type] Bit path

#0 SJR In synchronization establishment,
0: A machine coordinate difference between the master axis and slave axis is output at a time as command pulses (axis movements are performed without acceleration/deceleration).
1: Axis movements are executed with the feedrate of manual rapid traverse and the acceleration/deceleration after interpolation in rapid traverse.

> **NOTE**
> When the one-direction synchronization establishment function under axis synchronous control is used (bit 0 (SSO) of parameter No.8305 is set to 1), the machine coordinate difference for synchronization establishment is output as command pulses at a time, regardless of the setting of this parameter. Acceleration/deceleration is not applied to the axis movements in the one-direction synchronization establishment.

	#7	#6	#5	#4	#3	#2	#1	#0
8307								FSS

[Input type] Parameter input
[Data type] Bit type

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 FSS The slave axis of flexible synchronization control or the slave axis of superimposed control is:
0: Not used as the master axis of axis synchronous control.
1: Used as the master axis of axis synchronous control.

8311	Axis number of master axis in axis synchronous control

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to Number of controlled axes

Select a master axis in axis synchronous control. In the parameter for the slave axis, set the axis number of the master axis.

Example 1)
When one set of axis synchronous control is used:

When the master axis is the first axis (X-axis), and the slave axis is the third axis (Z-axis), set parameter No. 8311 as follows:

Parameter No.8311	X (first axis)	= 0
Parameter No.8311	Y (second axis)	= 0
Parameter No.8311	Z (third axis)	= 1
Parameter No.8311	A (fourth axis)	= 0

Example 2)

When three sets of axis synchronous control is used:
When the master axes are the first axis, second axis, and third axis, and the slave axes are the sixth axis, fifth axis, and fourth axis, set parameter No. 8311 as follows:

Parameter No.8311	X (first axis)	= 0
Parameter No.8311	Y (second axis)	= 0
Parameter No.8311	Z (third axis)	= 0
Parameter No.8311	A (fourth axis)	= 3
Parameter No.8311	B (fifth axis)	= 2
Parameter No.8311	C (sixth axis)	= 1

Example 3)

When the multiple slave axes of axis synchronous control are used in each path:
When the master axes are the first axis of the each path, and the slave axes are the fourth axis and fifth axis of the each path, set parameter No. 8311 as follows:

	Path-1		Path-2	
Parameter No.8311	X (first axis)	= 0	X (first axis)	= 0
Parameter No.8311	Y (second axis)	= 0	Y (second axis)	= 0
Parameter No.8311	Z (third axis)	= 0	Z (third axis)	= 0
Parameter No.8311	A (fourth axis)	= 1	A (fourth axis)	= 1
Parameter No.8311	B (fifth axis)	= 1	B (fifth axis)	= 1

8312	Enabling/disabling slave axis mirror image

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0, 100

When the slave axis mirror image is enabled on axis synchronous control, set this parameter to 100. If 0 is set in this parameter, the slave axis mirror image is disabled. The setting for the slave axis is available.

Example)

For reverse synchronization with the master axis being the third axis and the slave axis being the fourth axis, set parameter No. 8312 as follows:

Parameter No.8312	X (first axis)	= 0
Parameter No.8312	Y (second axis)	= 0
Parameter No.8312	Z (third axis)	= 0
Parameter No.8312	A (fourth axis)	= 100

NOTE

In synchronous operation with mirror image applied, synchronization error compensation, synchronization establishment, synchronization error checking, and correction mode cannot be used.

8314	Maximum allowable error in synchronization error check based on machine coordinates

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)

[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets a maximum allowable error in a synchronization error check based on machine coordinates. When the error between the master and slave axes in machine coordinates exceeds the value set in this parameter, the machine stops with the servo alarm SV0005, "SYNC EXCESS ERROR (MCN)".
The setting for the slave axis is available.

> **NOTE**
> Set 0 in this parameter when a synchronization error check is not made.

8323	Limit in positional deviation check in axis synchronous control

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999
This parameter sets the maximum allowable difference between the master axis and slave axis position deviations. When the absolute value of a positional deviation difference exceeds the value set in this parameter in axis synchronous control, the alarm DS0001, "SYNC EXCESS ERROR (POS DEV)" is issued.
The setting for the slave axis is available. If 0 is specified in this parameter, no position deviation difference check is made.

8325	Maximum compensation value in synchronization establishment based on machine coordinates

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the maximum compensation value for synchronization. When a compensation value exceeding the value set in this parameter is detected, the servo alarm SV0001, "SYNC ALIGNMENT ERROR" is issued, and the synchronization establishment is not performed.
The setting for the slave axis is available. To enable this parameter, set the bit 7 (SOF) of parameter No.8303 to 1. When 0 is set in this parameter, synchronization establishment is not performed.

8326	Difference between master axis and slave axis reference counters

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

The difference between the master axis reference counter and slave axis reference counter (master axis and slave axis grid shift) is automatically set when automatic setting for grid positioning is performed. Then, the difference is transferred together with an ordinary grid shift value to the servo system when the power is turned on. This parameter is set with a slave axis.

8327	Torque difference alarm detection timer

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] msec
[Valid data range] 0 to 4000

This parameter sets a time from the servo preparation completion signal, SA <Fn000.6>, being set to “1” until torque difference alarm detection is started in axis synchronous control.
When 0 is set in this parameter, the specification of 512 msec is assumed.
The setting for the slave axis is available.

8330	Multiplier for a maximum allowable synchronization error immediately after power-up

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 100

Until synchronization establishment is completed immediately after power-up, synchronization error excessive alarm 2 is checked using the maximum allowable error (parameter No. 8332) multiplied by the value set in this parameter.
If the result produced by multiplying the value of parameter No. 8332 by the value of this parameter exceeds 32767, the value is clamped to 32767.

8331	Maximum allowable synchronization error for synchronization error excessive alarm 1

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 32767

This parameter sets a maximum allowable synchronization error for synchronization error excessive alarm 1.
The setting for the slave axis is available.

8332	Maximum allowable synchronization error for synchronization error excessive alarm 2

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 32767

This parameter sets a maximum allowable synchronization error for synchronization error excessive alarm 2.
The setting for the slave axis is available.

8333	Synchronization error zero width for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 32767

When a synchronization error below the value set in this parameter is detected, synchronization error compensation is not performed.
The setting for the slave axis is available.

8334	Synchronization error compensation gain for each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 1 to 1024

This parameter sets a synchronization error compensation gain.
Compensation pulses found by the following expression are output for the slave axis:

Compensation pulses = Synchronization error × (Ci/1024)

Ci: Compensation gain

The setting for the slave axis is available.

8335	Synchronization error zero width 2 for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 32767

This parameter sets synchronization error zero width 2 for synchronization error smooth suppression.
The setting for the slave axis is available.

> **NOTE**
> Set a value less than the value set in parameter No. 8333.

8336	Synchronization error compensation gain 2 for each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1024

This parameter sets synchronization error compensation gain 2 for synchronization error smooth suppression.
The setting for the slave axis is available.

> **NOTE**
> Set a value less than the value set in parameter No. 8334.

8337	M code for turning off synchronization in axis synchronous control
8338	M code for turning on synchronization in axis synchronous control

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 999999999

This parameter specifies an M code for switching between synchronous operation and normal operation.
The M code set in this parameter is not buffered.

> ⚠ **CAUTION**
> To switch between synchronous operation and normal operation, specify the M code set in parameter No. 8337 or 8338.

1.6.14 Diagnosis Data

The synchronization error and compensation are displayed on the diagnostic screen.

3500	Synchronization error for each axis

[Unit of data] Detection unit
[Description] The difference between the positions of the master and slave axes (synchronization error) is displayed.
It is displayed for the axis number of the slave axis.

3501	Synchronization error compensation for each axis

[Unit of data] Detection unit
[Description] The total number of compensation pulses output to the slave axis (synchronization error compensation) is displayed. This number is displayed for the axis number of the slave axis.

1.6.15 Alarm and Message

Number	Message	Description
PS0213	ILLEGAL COMMAND IN SYNCHRO-MODE	In axis synchronous control, the following errors occurred during the synchronous operation. 1) The program issued the move command to the slave axis. 2) The manual operation (jog feed or incremental feed) is performed to the slave axis. 3) The program issued the automatic reference position return command without specifying the manual reference position return after the power was turned on.
PS5379	WRITE PROTECTED TO SLAVE AXIS	It is not possible to directly set the parameters for the slave axis under axis synchronous control.
PS5580	CANNOT GET SYSTEM VARIABLE	System variable (#180801 to #180850) cannot be read when parameter No.8311 is incorrect value. Confirm the setting of parameter No.8311.
DS0001	SYNC EXCESS ERROR (POS DEV)	In axis synchronous control, the difference in the amount of positional deviation between the master and slave axes exceeded the parameter No. 8323 setting value. This alarm occurs for the master or slave axis.

Number	Message	Description
DS0002	SYNC EXCESS ERROR ALARM 1	In axis synchronous control, the difference in the amount of synchronization between the master and slave axes exceeded the parameter No. 8331 setting value. This alarm occurs only for the slave axis.
DS0003	SYNCHRONIZE ADJUST MODE	The system is in the synchronize adjust mode.
SV0001	SYNC ALIGNMENT ERROR	In axis synchronous control, the amount of compensation for synchronization exceeded the parameter No. 8325 setting value. This alarm occurs for the master or slave axis.
SV0002	SYNC EXCESS ERROR ALARM 2	In axis synchronous control, the amount of synchronization error exceeded the parameter No. 8332 setting value. When the synchronization is not completed after power-up, the determination is made by the parameter value No. 8332 multiplied by the parameter No. 8330 multiplier. This alarm occurs only for a slave axis only.
SV0005	SYNC EXCESS ERROR (MCN)	In axis synchronous control, for synchronization, the difference value of the machine coordinate between a master and slave axes exceeded the parameter No. 8314 setting value. This alarm occurs for a master or slave axis.
SV0420	SYNC TORQUE EXCESS	In axis synchronous control, for synchronization, the difference value of torque between a master and slave axes exceeded the parameter No. 2031 setting value. This alarm occurs for a master axis.

1.6.16 Caution

Caution

⚠ **CAUTION**

1. When making a synchronization error check, ensure that the reference position on the master axis and the reference position on the slave axis must be at the same position.
2. In manual reference position return operation, the same operation is performed along the master axis and slave axis until a deceleration operation starts. After a deceleration operation starts, grid detection is performed for the master axis and slave axis independently of each other.
3. Pitch error compensation and backlash compensation are performed for the master axis and slave axis independently of each other.

Note

NOTE

1 During axis synchronous control, a movement based on the reference position return check (G27), automatic reference position return (G28), 2nd/3rd/4th reference position return (G30), or machine coordinate system selection (G53) command is made as described below according to the setting of bit 2 (SRF) of parameter No. 8305.
   <1> When SRF = 0, the same movement as made along the master axis is made along the slave axis.
   <2> When SRF = 1, a movement is made along the slave axis to the specified position independently of a movement made along the master axis to the specified position.

2 A command not involving a movement along an axis such as the workpiece coordinate system setting command (G92) and local coordinate system setting command (G52) is set with the master axis according to the master axis programming.

3 During synchronous operation, the axis-by-axis signals such as for external deceleration, interlock, and machine lock are enabled for the master axis only. During synchronous operation, those signals for the slave axis are ignored.

4 When switching the synchronization state in a program, be sure to specify M codes (parameters Nos. 8337 and 8338) for turning synchronization on and off. By switching between the input signals SYNC <Gn138> and SYNCJ <Gn140> from the PMC with the M codes, the synchronization state can be switched in the program.

5 When controlled axis detach is performed, the synchronization state is cancelled. When performing controlled axis detach, perform detach for the master axis and slave axis at the same time.

6 If a programmed command is specified for the slave axis during synchronous operation, an alarm PS0213, “ILLEGAL COMMAND IN SYN-CHRO-MODE” is issued.
   A programmed command can be specified for the slave axis when switching between synchronous operation and normal operation is set to 0 (with bit 5 (SCAx) of parameter No. 8304 set to 0) and the signal SYNC/SYNCJ select normal operation.

7 Axis synchronous control and PMC axis control cannot be used at the same time.

8 When bit 4 (SYPx) of parameter No. 8303 is changed from 0 to 1 to use automatic slave axis parameter setting, those slave axis parameters that have already been set are not automatically set. Automatic slave axis parameter setting is enabled after parameter No. 8311 and bit 4 (SYPx) of parameter No. 8303 are set.

9 If inputting a parameter file after enabling automatic slave axis parameter setting, set parameter No. 8311 and bit 4 (SYPx) of parameter No. 8303 for both the CNC and the parameter file before inputting it.

10 If the manual handle interrupt and the tool retract and recover is performed on the axes of axis synchronous control, set “1” to signal SYNCJ.

> **NOTE**
> 11 When the machine lock is effective, the coordinate of master axis is as follows.
> - The absolute and relative coordinate are updated.
> - The machine coordinate is not updated.
>
> And, the absolute, relative and machine coordinate of slave axis are not updated.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Axis synchronous control

1.7 TANDEM CONTROL

Overview

If a single motor cannot produce sufficient torque to move a large table, for example, this function allows two motors to be used. By means of this function, two motors can be used to perform movement along a single axis.

Positioning is carried out only for the master axis. The slave axis is used only to produce a torque. By means of this function, double the amount of torque can be obtained.

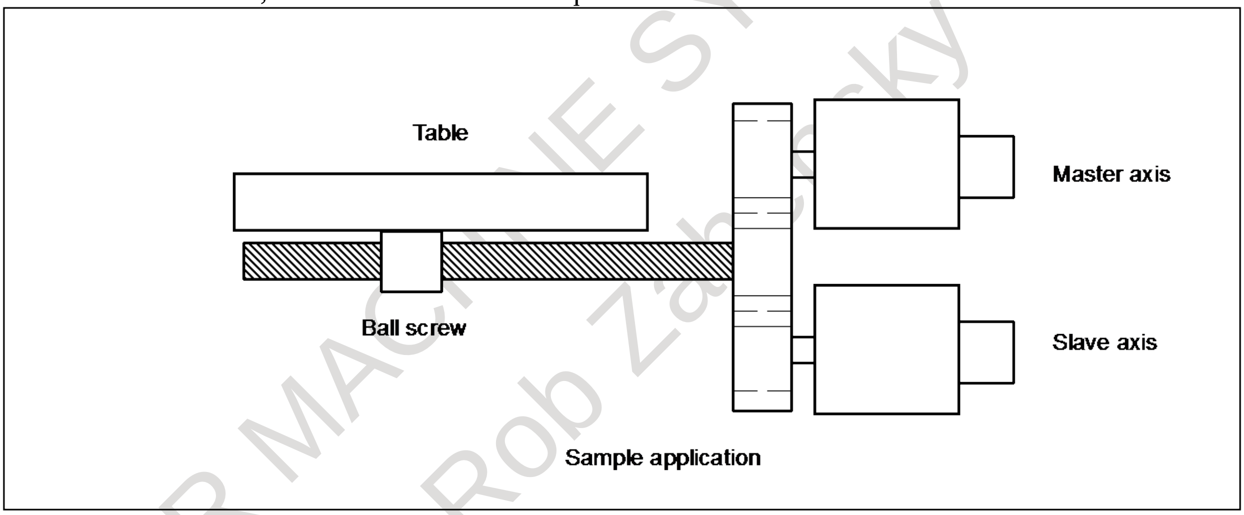


Fig. 1.7 (a)

The CNC generally processes the two axes of tandem control as a single axis. In the management of servo parameters and the monitoring of servo alarms, however, the two axes are handled individually.

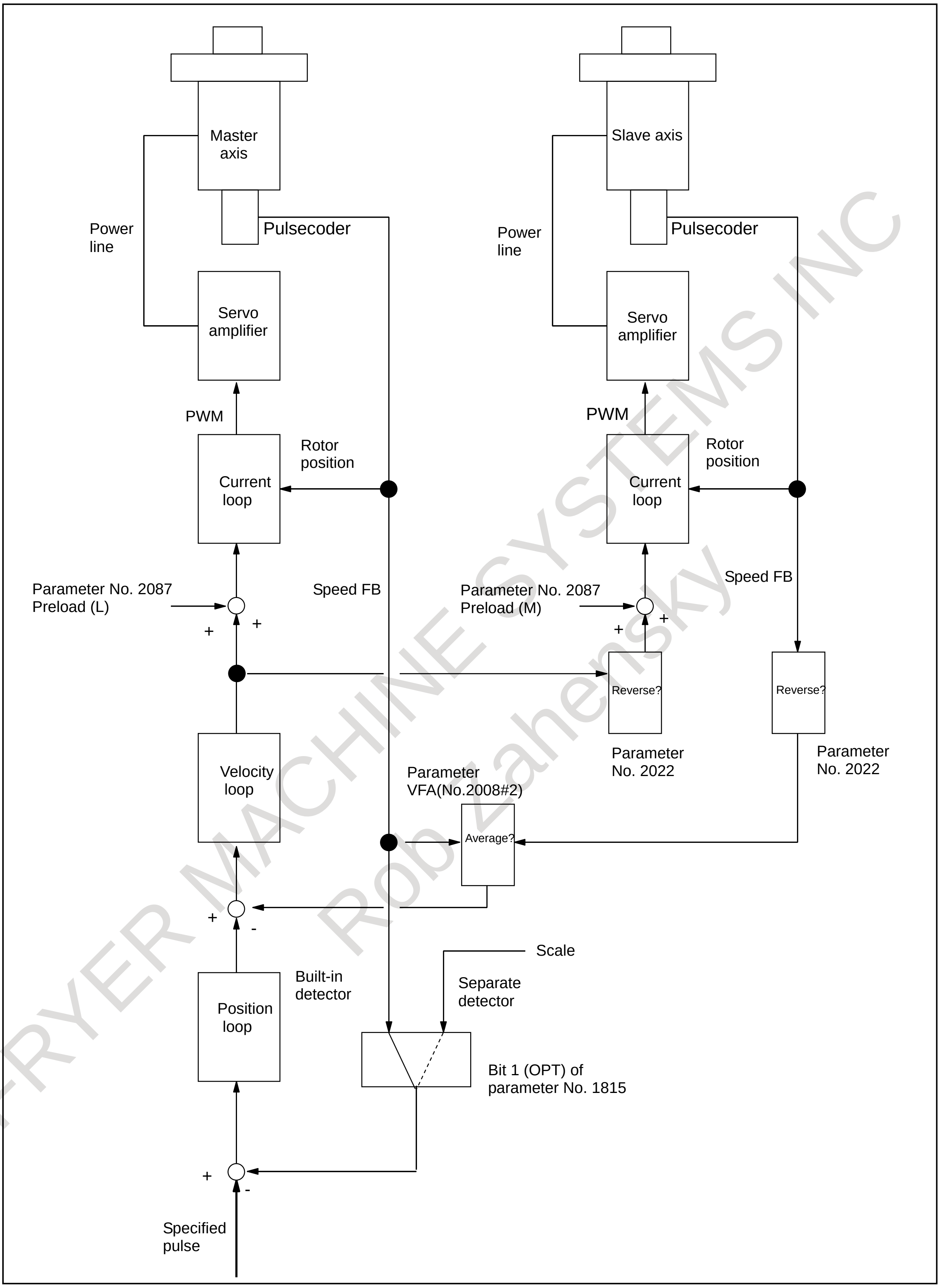


Fig. 1.7 (b) Block Diagram of Tandem Control

Explanation

- Axis configuration in tandem control

To specify the axis configuration in tandem control, follow the procedure below:

(1) Tandem control can be performed for up to sixteen pairs of axes.
It can be performed for up to twelve pairs of axes for each path.

(2) In terms of controlled axes, the pair of axes is handled as two separate axes. For a programmed command or manual feed, the pair of axes is handled as a single axis.

(3) The pair of axes is handled as two separate axes in the management of servo parameters and the monitoring of servo alarms.

(4) Assign two consecutive numbers, that is one odd and one even number, to the master and slave axes as their servo axis numbers (parameter No. 1023). Assign the smaller number to the master axis.
(Example)
If the servo axis number of the master axis (parameter No. 1023) is set to 1, specify servo axis number 2 for the corresponding slave axis. If the servo axis number of the master axis is set to 3, specify servo axis number 4 for the corresponding slave axis.
In terms of controlled axis order, it is necessary to set master axis ahead of slave axis.

(5) The master and slave axes may have the same name or different names.

(6) A subscript can be attached to an axis name like X1, X2, XM, and XS. If the same axis name is used for multiple axes, and a unique subscript is assigned to each of those axes, the axes can be distinguished from each other on the screen display, or which of those axes issued an alarm can be identified.
Set a subscript in parameter No. 3131.

(7) The slave axis is handled as a controlled axis. Set bit 0 (NDPx) of parameter No. 3115 to 1 to suppress the position display.
The following sample axis configuration is for a machine with five axes X, Y, Z, A, and B. The X-axis and Y-axis are the master axes of tandem control.
Number of controlled axes = Seven

Table 1.7 (a)

Axis number	Displayed axis name	Axis name (No.1020)	Subscript (No.3131)	Servo axis number (No.1023)	Tandem axis (No.1817#6)	
1	XM	88	77	1	1	Master axis of tandem control
2	XS	88	83	2	1	Slave axis of tandem control
3	Z	90	0	5	0	
4	A	65	0	6	0	
5	B	66	0	9	0	
6	YM	89	77	3	1	Master axis of tandem control
7	YS	89	83	4	1	Slave axis of tandem control

(8) The master and slave axes must be included in the same path.

(9) Set an absolute position detector only on the master axis. If it is set on the slave axis, alarm SV0006 "ILLEGAL TANDEM AXIS" is issued.

- Preload function

By adding an offset to the torque controlled by the position (velocity) feedback device, the function can apply opposite torques to the master and slave axes so that equal and opposite movements are performed for both axes. This function can reduce the effect of backlash on the master and slave axes caused by the tandem connection of the two motors via a gear. This function, however, cannot reduce backlash between the ball screw and table or other backlash inherent to the machine.

If a preload of X is set for the master axis and -X for the slave axis, the opposing preload torques are continuously applied to the two axes, even at rest, as shown below (Fig. 1.7 (c)):

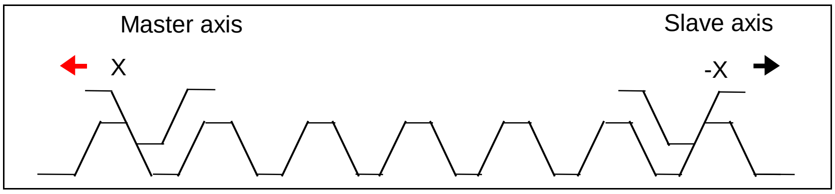


Fig. 1.7 (c)

> ⚠ **CAUTION**
> 1 Specify as low a preload as possible. Avoid specifying a preload higher than the rated torque. Too high a preload will trigger an overload alarm because the specified torques continue to be applied, even at rest. A preload that is only slightly higher than the frictional force is recommended. Thus, the recommended preload may be about one-third of the rated torque.
> 2 If the motors rotate in opposite directions (different signs are specified in parameter No. 2022), specify the preload values with the same sign.

- Velocity feedback average function

As shown in the block diagram of tandem control, the motor of the slave axis is not subject to velocity control. A machine with a large amount of backlash may become unstable if the motor of the slave axis vibrates as a result of backlash in the gear. This can be overcome by applying velocity control to the slave axis also. This velocity feedback average function is enabled when bit 2 of parameter No. 2008 is set to 1.

- Improved stability of a closed-loop system

The following two functions can increase the stability and position gain of a closed-loop system having a linear scale:

- Dual position feedback function
- Machine velocity feedback function

For details of these functions, refer to "FANUC AC SERVO MOTOR α*i*/β*i* series, LINEAR MOTOR L*i*S series, SYNCHRONOUS BUILT-IN SERVO MOTOR D*i*S series PARAMETER MANUAL (B-65270EN)".

- Notes on stability of tandem control

An important factor affecting stability in tandem control is the capability of back feed. Back feed is to cause movement along either the master or slave axis from the other axis, via the transmission mechanism connecting the two axes. A machine without this capability may be inclined to become unstable and require adjustments.

- Connection of axis signals

The DI/DO signals, generally connected to each axis, must be connected only to the master axis of two axes of tandem control. The signals need not be connected to the slave axis. The following signals, however, may have to be connected depending on the application.

i) Controlled axis detach signals DTCH1 to DTCH8<Gn124> and servo off signals SVF1 to SVF8<Gn126>
   Connect these signals so that the master and slave axis signals are simultaneously input.

ii) Overtravel signals *+L1 to *+L8<Gn114>, *-L1 to *-L8<Gn116>
   Connect the signal so that 1 is always output as the overtravel signal for the slave axis.

If the slave axis stroke limit must also be detected, connect the signals so that the signal detected on the slave axis is sent to the overtravel signal of the master axis.

- Connecting motors

Connect the motors according to the servo axis numbers. Connect the feedback cable of the slave axis.
(Sample connection for position feedback cable)

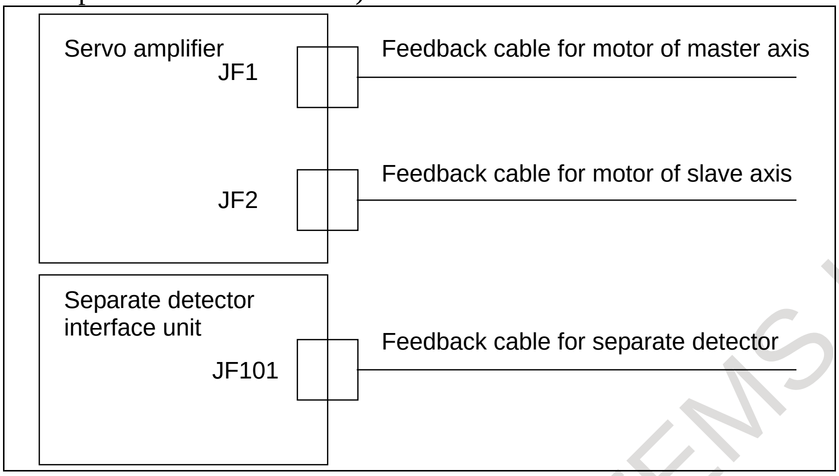


Fig. 1.7 (d)

- Servo alarms

Motor overload and other servo alarms are displayed separately for the master and slave axes.

- Servo off of slave axis

Specification of servo off of slave axis depends on the parameter TSF (No.11803#2) as follows.

- In case parameter TSF=0
  The servo of the slave axis is turned off together with that of the master axis.
- In case parameter TSF=1
  The servo of the slave axis is turned off independently of, rather than together with, that of the master axis.

NOTE

1. Use the parameter TSF for the slave axis.
2. Specify the parameter TSF when both the master and slave axes are at a stop.
3. Setting the parameter TSF to 1 requires consideration on the ladder side, because the servo of the slave axis is turned off independently of, rather than together with, that of the master axis.

Parameter

- Setting data (parameters)

The parameters that are generally set for each axis can, when set for axes under tandem control, be classified into the following three groups:

i) Parameters in which identical values must be set for the master and slave axes
ii) Parameters that must be specified only for the master axis (The corresponding parameter for the slave axis is not used.)
iii) Parameters for which different values may be set for the master and slave axes

The classifications of the parameters are described below. Any parameter that is not listed in the tables for the three classifications should be processed as a parameter of type i) and, specify identical values for the master and slave axes.

⚠ **CAUTION**

Note that, if different values are set for the master and slave axes in a parameter of type i), the operations for the two axes of tandem control will not be performed correctly.

- Care must be taken to specify the following two servo parameters, according to the directions of rotation around the master and slave axes.
  No. 2022 Direction of rotation of the motor
  No. 2087 Preload value
  In parameter No. 2022, specify 111 for forward rotation and -111 for the reverse rotation.
  In parameter No. 2087, specify values having identical signs when the motors of the master and slave axes rotate in opposite directions. Specify values having different signs when the motors of the master and slave axes rotate in the same direction.
- If a separate detector is used, use of the separate detector must be set for the master axis. For the slave axis, use of a built-in Pulsecoder must be set. Therefore, pay particular attention to setting the following parameters.
  No. 1815#1 Separate detector
  No. 2024 Number of position pulses (PPLS)
  No. 1821 Capacity of a reference counter
  No. 2084 Flexible feed gear ratio (numerator)
  No. 2085 Flexible feed gear ratio (denominator)

If, for example, a motor with αi Pulsecoder is used with a linear scale capable of detecting a position in 1-μm units, and if a single rotation of the motor produces a movement of 4 mm, specify the parameters as shown below:

	Master axis	Slave axis
No. 1815#1=	1	0
No. 2024 =	4000	12500
No. 1821 =	4000	4000
No. 2084 =	1	4
No. 2085 =	1	1000

- Parameters that should be set to the same values for the master and slave axes

Parameter No.	Meaning of parameters
1005#0	Reference position return
1005#1	Setting of the reference position without dogs
1006#0 to #1	Linear / rotary axis
1006#3	Diameter/radius specification
1006#5	Direction of manual reference position return
1013#0 to #3	Setting unit
1020	Program axis name
1025	Program axis name 2 for each axis
1026	Program axis name 3 for each axis
1240	Coordinate value on the 1st reference point of the machine coordinate system
1241	Coordinate value on the 2nd reference point of the machine coordinate system
1242	Coordinate value on the 3rd reference point of the machine coordinate system
1243	Coordinate value on the 4th reference point of the machine coordinate system
1260	Move distance per rotation of rotary axis
1420	Rapid traverse rate
1421	F0 speed of rapid traverse override
1620	Time constant of rapid traverse linear acceleration/deceleration
1621	Time constant of rapid traverse bell shaped acceleration/deceleration
1622	Time constant of acceleration/deceleration in cutting feed
1623	FL speed of acceleration/decelerati after interpolation in cutting feed
1624	Time constant acceleration/deceleration in jog feed
1625	FL speed of acceleration/deceleration in jog feed
1626	Time constant of during thread cutting cycle
1627	FL speed of acceleration/deceleration during thread cutting cycle
1820	Command multiplier (CMR)
18XX	Most of parameters related to digital servo
20XX	Most of parameters related to digital servo

- Parameters that should be set only for the master axes

Parameter No.	Meaning of parameters
0012#0	Mirror image
0012#7	Controlled axis detach
1004#7	Input unit 10 times
1005#4	External deceleration in positive direction
1005#5	External deceleration in negative direction
1005#7	Servo control off
1022	Basic / parallel axis specification
1220	External workpiece zero point offset
1221	Workpiece zero point offset of workpiece coordinate system 1 (G54)
1222	Workpiece zero point offset of workpiece coordinate system 2 (G55)
1223	Workpiece zero point offset of workpiece coordinate system 3 (G56)
1224	Workpiece zero point offset of workpiece coordinate system 4 (G57)
1225	Workpiece zero point offset of workpiece coordinate system 5 (G58)
1226	Workpiece zero point offset of workpiece coordinate system 6 (G59)
1423	Jog feedrate
1424	Manual rapid traverse
1425	FL rate in manual reference position return
1427	External deceleration rate at rapid traverse
1430	Maximum cutting feedrate
1815#5	Absolute Pulsecoder
2008#2	Velocity feedback average function
19655	Axis number of the linear axis to which a rotation axis belongs
19658	Angular displacement of a rotary axis
19659	Offset value for angular displacement of a rotation axis
19660	Origin offset value of a rotation axis
19667	Control point shift vector

- Parameters that may be set to different values for the master and slave axes

Parameter No.	Meaning of parameters
1023	Servo axis number
1310#0	Stored stroke check 2
1310#1	Stored stroke check 3
1320	Coordinate value I of stored stroke check 1 in the positive direction
1321	Coordinate value I of stored stroke check 1 in the negative direction
1322	Coordinate value of stored stroke check 2 in the positive direction
1323	Coordinate value of stored stroke check 2 in the negative direction
1324	Coordinate value of stored stroke check 3 in the positive direction
1325	Coordinate value of stored stroke check 3 in the negative direction
1326	Coordinate value II of stored stroke check 1 in the positive direction
1327	Coordinate value II of stored stroke check 1 in the negative direction
1815#1	Separatedetector
1821	Arbitrary reference counter capacity
2022	Motor rotation direction
2024	Number of position pulses (PPLS)
2084	Numerator of flexible feed gear ratio
2085	Denominator of flexible feed gear ratio
2087	Preload value
3115#0	Current position display

1020	Program axis name for each axis

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 65 to 67, 85 to 90

An axis name (axis name 1: parameter No. 1020) can be arbitrarily selected from A, B, C, U, V, W, X, Y, and Z. (When G code system A is used with the lathe system, however, U, V, and W are not selectable.) When bit 0 (EEA) of parameter No. 1000 is set to 1, the length of an axis name can be extended to three characters by setting axis name 2 (parameter No. 1025) and axis name 3 (parameter No. 1026) (extended axis name).
For axis names 2 and 3, a character from 0 to 9 and A to Z of ASCII code can be arbitrarily selected. However, the setting of axis name 3 for each axis is invalid if axis name 2 is not set. Moreover, if a character from 0 to 9 is set as axis name 2, do not use a character from A to Z as axis name 3.

(Tip) ASCII code

Axis name	X	Y	Z	A	B	C	U	V	W
Setting	88	89	90	65	66	67	85	86	87

When G code system A is used with the lathe system, and the character X, Y, Z, or C is used as axis name 1 of an axis, a command with U, V, W, or H specified for axis name 1 represents an incremental programming for the axis.

NOTE
1. When a multiple repetitive canned cycle for turning is used, no character other than X, Y, and Z can be used as the address of the axis.
2. When the custom macro function is enabled, the same extended axis name as a reserved word cannot be used. Such an extended axis name is regarded as a reserved word.
   Because of reserved words of custom macros, extended axis names that start with the following two characters cannot be used:
   AB, AC, AD, AN, AS, AT, AX, BC, BI, BP, CA, CL, CO, US, WH, WR, XO, ZD, ZE, ZO, ZW
3. In a macro call, no extended axis name can be used as an argument.

1023	Number of the servo axis for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 80

This parameter associates each control axis with a specific servo axis. Specify values 1+8n, 2+8n, 3+8n, 4+8n, 5+8n, and 6+8n (n = 0, 1, 2, …, 9) like 1, 2, 3, 4, 5, …, 77, and 78.
The control axis number is the order number that is used for setting the axis-type parameters or axis-type machine signals

- With an axis for which Cs contour control/spindle positioning is to be performed, set -(spindle number) as the servo axis number.
  Example)
  When exercising Cs contour control on the fourth controlled axis by using the first spindle, set -1.
- For tandem controlled axes or electronic gear box (EGB) controlled axes, two axes need to be specified as one pair. So, make a setting as described below.

Tandem axis: For a master axis, set an odd (1, 3, 5, 9, ...) servo axis number. For a slave axis to be paired, set a value obtained by adding 1 to the value set for the master axis.

EGB axis: For a slave axis, set an odd (1, 3, 5, 9, ...) servo axis number. For a dummy axis to be paired, set a value obtained by adding 1 to the value set for the slave axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1815							OPTx	

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#1 OPTx The separate position detector is:
0: Not to be used (semi-closed system)
1: To be used (full-closed system)

> **NOTE**
> 1 In case of using the absolute position detector (bit 5 (APCx) of parameter No.1815 is set to 1), please set the following parameters that correspond to the absolute position detector which is actually used.
> - parameter No.1815#6, No.1815#0, No.1817#3, No.1868, No.2275#1, No.2394
>
> If these parameters are not set correctly, the machine coordinates are not correctly established at power-on.
> 2 Set this parameter to 1 when using a linear scale with reference marks or a linear scale with distance-coded reference marks (serial) (full-closed system).

	#7	#6	#5	#4	#3	#2	#1	#0
1817		TANx						

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#6 TANx Tandem control
0: Not used
1: Used

> **NOTE**
> Set this parameter to both master axis and slave axis.

1821	Reference counter size for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

Set a reference counter size.
As a reference counter size, specify a grid interval for reference position return based on the grid method.
When a value less than 0 is set, the specification of 10000 is assumed.
When a linear scale with absolute address reference marks is used, set the interval of mark 1.

	#7	#6	#5	#4	#3	#2	#1	#0
2008						VFA		

[Input type] Parameter input
[Data type] Bit axis

#2 VFA In tandem control, the speed feedback average function is:
0: Disabled.
1: Enabled.

2021	Load inertia ratio

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 32767

(Load inertia)/(motor inertia) × 256

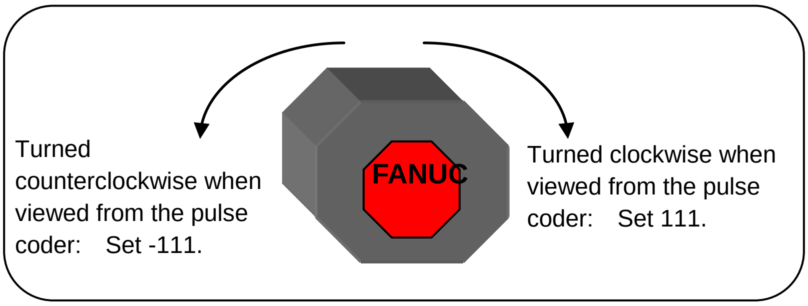
For tandem control:
(Load inertia)/(motor inertia) × 256/2
Set the same value for the master axis and slave axis.

2022	Direction of motor rotation

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] -111,111

Set the direction of motor rotation.
If the motor turns clockwise when viewed from the Pulsecoder side, set 111. If the motor turns counterclockwise when viewed from the Pulsecoder side, set -111.
When the master axis and slave axis rotate in opposite directions each other, this parameter is used for switching.



2087	Preload value for each axis (Tcmd offset)

[Input type] Parameter input
[Data type] Word axis
[Unit of data] (Ampere limit)/7282
[Valid data range] -1821 to 1821

An offset is applied to a torque command to suppress backlash.
Set a value much greater than the friction.
As a guideline, specify a value that is about one-third of the rated torque.

[Example] To set a torque equivalent to 3 A in the opposite directions:
When the ampere limit is 40 A

3/(40/7282) = 546
Master side = 546
Slave side = -546

	#7	#6	#5	#4	#3	#2	#1	#0
3115								NDPx

[Input type] Parameter input
[Data type] Bit axis

#0 NDPx The current position is:
0: Displayed.
1: Not displayed.

> **NOTE**
> When using the electronic gear box (EGB) function, set 1 for the EGB dummy axis to disable current position display.

3131	Subscript of axis name

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 9, 65 to 90

In order to distinguish axes under parallel operation, synchronization control, and tandem control, specify a subscript for each axis name.

Setting value	Meaning
0	Each axis is set as an axis other than a parallel axis, synchronization control axis, and tandem control axis.
1 to 9	A set value is used as a subscript.
65 to 90	A set letter (ASCII code) is used as a subscript.

[Example] When the axis name is X, a subscript is added as indicated below.

Setting value	Axis name displayed on a screen such as the position display screen
0	X
1	X1
77	XM
83	XS

If a multi-path system is used, no extended axis name is used within a path, and no subscript is set for the axis names, then the path number is automatically used as the subscript for the axis names. To disable the display of axis name subscripts, set a blank (32) of ASCII code in the parameter for specifying an axis name subscript.

NOTE

If even one axis in a path uses an extended axis name when bit 2 (EAS) of parameter No. 11308 is set to 0, subscripts cannot be used for axis names in the path.

	#7	#6	#5	#4	#3	#2	#1	#0
11803						TSF		

[Input type] Parameter input
[Data type] Bit axis

#2 TSF Under tandem control, the servo of the slave axis is turned off:
0: Together with that of the master axis.
1: Independently of that of the master axis.

NOTE

1 Use this parameter for the slave axis under tandem control.
2 Specify this parameter when both the master and slave axes under tandem control are at a stop.
3 Setting this parameter to 1 requires consideration on the ladder side, because the servo of the slave axis is turned off independently of, rather than together with, that of the master axis.

Alarm and message

Number	Message	Description
SV0006	ILLEGAL TANDEM AXIS	For the slave axis under tandem control, absolute position detection is set (bit 5 (APCx) of parameter No. 1815 = 1).
SV1055	ILLEGAL TANDEM AXIS	The setting of parameter No. 1023 and bit 6 (TDMx) of parameter No.1817 is invalid for tandem control.

NOTE

⚠ CAUTION

In case the names of the master axis and slave axis are different, be sure not to specify a programmed command for the slave axis.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Tandem control

1.8 ANGULAR AXIS CONTROL

Overview

When the angular axis installed makes an angle other than 90° with the perpendicular axis, the angular axis control function controls the distance traveled along each axis according to the inclination angle as in the case where the angular axis makes 90° with the perpendicular axis.

Arbitrary axes can be specified as a set of an angular axis and perpendicular axis by parameter setting.

The actual distance traveled is controlled according to an inclination angle. However, a program, when created, assumes that the angular axis and perpendicular axis intersect at right angles. The coordinate system used at this time is referred to as the program coordinate system. (The program coordinate system may be referred to as the Cartesian coordinate system, and the actual move coordinate system may be referred to as the slanted coordinate system or machine coordinate system.)

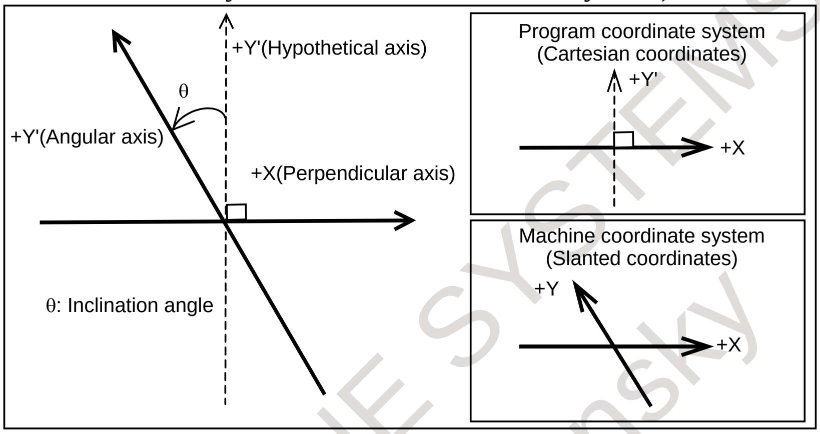


Fig. 1.8 (a)

Explanation

- Travel distance of each axis

When the amounts of travel along the angular axis and the perpendicular axis are Ya and Xa, respectively, the amounts are controlled according to the formulas shown below.

$$Ya = \frac{Yp}{\cos\theta}$$

Xa,*Ya*: Actual distance
Xp,*Yp*: Programmed distance

The amount of travel along the perpendicular axis is corrected by the influence of travel along the angular axis, and is determined by the following formula:

$$Xa = Xp - C \times Yp \times \tan\theta$$

> **NOTE**
> The coefficient C is 2 in the case of diameter specification for the perpendicular axis (X) or 1 in the case of radius specification.

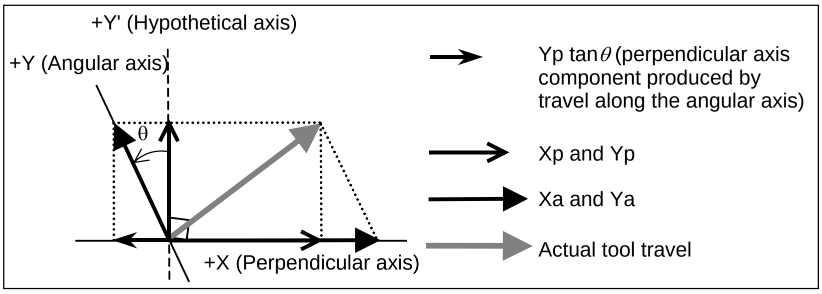


Fig. 1.8 (b)

- Feedrate

When the Y-axis is an angular axis, and the X-axis is a perpendicular axis, the feedrate along each axis is controlled as described below so that the feedrate in the tangent direction becomes Fp.
The feedrate component along the Y-axis is determined by the following expressions:

$$Fay = \frac{Fp}{\cos\theta}$$

Fa represents the actual feedrate.
Fp represents a programmed feedrate.

Fax = Fp – Fp × tanθ

- Absolute and relative position display

An absolute and a relative position are indicated in the programmed Cartesian coordinate system.

- Machine position display

A machine position indication is provided in the machine coordinate system where an actual movement is taking place according to an inclination angle.

Method of use

The angular and perpendicular axes for which angular axis control is to be applied must be specified beforehand, using parameters Nos. 8211 and 8212. When 0 is set in one of the parameters, the same number is specified in the parameters, or a number other than the controlled axis numbers is specified in a parameter, however, an angular axis and perpendicular axis are selected according to the Table 1.8 (a).

Table 1.8 (a)

	Angular axis	Perpendicular axis
M series	Y-axis of the basic three axes (axis with 2 set in parameter No. 1022)	Z-axis of the basic three axes (axis with 3 set in parameter No. 1022)
T series	X-axis of the basic three axes (axis with 1 set in parameter No. 1022)	Z-axis of the basic three axes (axis with 3 set in parameter No. 1022)

- Bit 0 (AAC) of parameter No. 8200 enables or disables the angular axis control. If the function is enabled, the distance traveled along each axis is controlled according to an angular angle parameter No. 8210.
- By using bit 2 (AZR) of parameter No. 8200, whether to make a movement along the perpendicular axis by a movement made along the angular axis when a manual reference position return operation is performed along the angular axis can be chosen. When a movement along the perpendicular axis is enabled (AZR = 1), a reference position return operation along the perpendicular axis can be performed by a movement made along the angular axis.
- Bit 3 (AZP) of parameter No. 8200 can be used to set reference position return end signals for the perpendicular axis ZP1 to ZP8<Fn094.0 to Fn094.7> to “0” when a movement is made along the perpendicular axis due to a movement along the angular axis.
- By setting the signal for disabling angular axis control for the perpendicular axis NOZAGC<Gn063.5> to “1”, slanted axis control only for the angular axis can be available. In this time the angular axis are converted to those along the slanted coordinate system without affecting commands to normal axis.
  Use this signal when operating each axis independently.

- Manual reference position return operation

A movement is made to the reference position (machine position) set in parameter No. 1240. By using bit 2 (AZR) of parameter No. 8200, whether to make a movement along the perpendicular axis when a reference position return operation is performed along the angular axis can be chosen.

- Automatic reference position return operation (G28, G30)

A movement to the middle point along the angular axis affects a movement along the perpendicular axis. It is possible to select between Cartesian coordinate system operation (compatible with FS16*i*) and

angular coordinate system operation as the movement along the angular axis from the intermediate position to the reference position, using bit 0 (ARF) of parameter No. 8209.
If manual reference position return operation is not performed even once after the power is turned on, operation is performed in the same sequence as for manual reference position return operation. So, specify commands first for the angular axis then for the perpendicular axis.
Example 1)
When the Y-axis is an angular axis and the X-axis is a perpendicular axis
(1) If the angular axis is first specified then the perpendicular axis is specified, reference position return operation is performed normally.
G28Y_;
G28X_;
(2) If the perpendicular axis is first specified then the angular axis is specified, or if the perpendicular axis and the angular axis are specified at the same time, alarm PS0372 is issued when a movement is made along the perpendicular axis.
G28X_; or G28X_Y_;
G28Y_;

Example 2)
Automatic reference position return example
(If the Y-axis is an angular axis, the X-axis is a perpendicular axis, and the angular angle is -30°)
<1> Automatic reference position return command on the X-axis from point P2
>G90G28X200.0
<2> Automatic reference position return command on the Y-axis from point P1
>G90G28Y100.0
(1) If bit 0 (ARF) of parameter No. 8209 is 1 (compatible with FS16*i*)
<1> Coordinates at P1

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	57.735
Y	100.000	Y	115.470

<2> Coordinates at P0

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	0.000
Y	0.000	Y	0.000

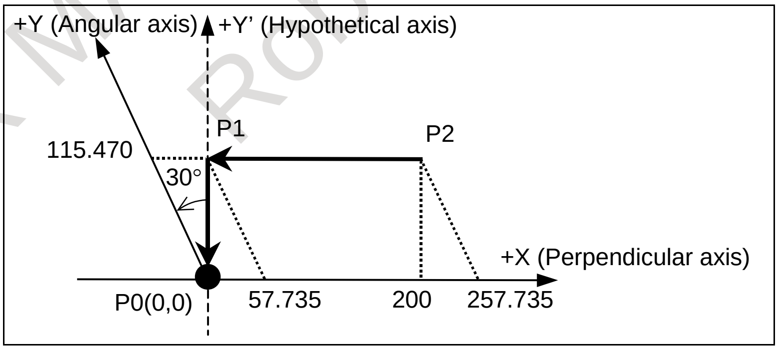


Fig. 1.8 (c)

(2) If bit 0 (ARF) of parameter No. 8209 is 0
<1> Coordinates at P1

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	0.000
Y	100.000	Y	115.470

<2> Coordinates at P0

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	0.000
Y	0.000	Y	0.000

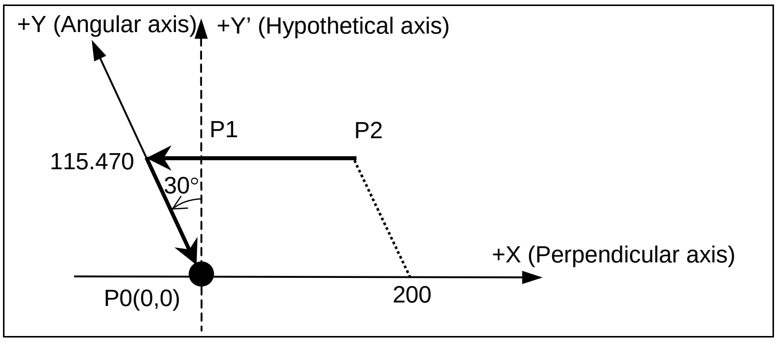


Fig. 1.8 (d)

- Reference position return operation of high-speed type

When a reference position is already established and a reference position return operation of high-speed type is to be performed, the reference position return operation need not be performed in the order from the angular axis to the perpendicular axis.

- Machine coordinate selection (G53)

By specifying (G90)G53X_Y_: (when the Y-axis is an angular axis, the X-axis is a perpendicular axis, and the inclination angle is -30°), a movement is made by rapid traverse.
However, a movement along the angular axis (G53 command) does not affect a movement along the perpendicular axis, regardless of whether the signal for disabling angular axis control for the perpendicular axis (NOZAGC) is turned on or off.
Example)

<1> Move command for movement from point P0 to point P1
>G90G53Y100.0

<2> Move command for movement from point P1 to point P2
>G90G53X200.0

<1> Coordinates of P1

(Absolute coordinate)		(Machine coordinate)	
X	-50.000	X	0.000
Y	86.603	Y	100.000

<2> Coordinates of P2

(Absolute coordinate)		(Machine coordinate)	
X	150.000	X	200.000
Y	86.603	Y	100.000

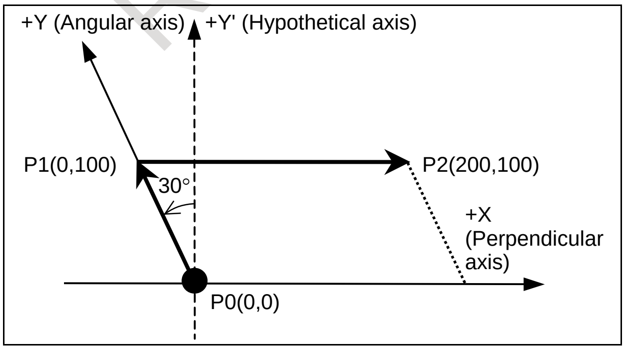


Fig. 1.8 (e)

- Commands for linear interpolation and positioning of linear interpolation type (G01, G00)

The tool moves to a specified position in the Cartesian coordinate system when the following is specified:
(G90)G00X_Y_; (when the Y-axis is an angular axis, the X-axis is a perpendicular axis, and the inclination angle is -30°)
or

(G90)G01X_Y_F_; (when the Y-axis is an angular axis, the X-axis is a perpendicular axis, and the inclination angle is -30°)

Example) Examples of positioning

<1> Move command for movement from point P0 to point P1

>G90G00Y100.0

<2> Move command for movement from P1 to P2

>G90G00X200.0

(1) When the signal for disabling angular axis control for the perpendicular axis (NOZAGC) is set to “0”

<1> Coordinates of P1

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	57.735
Y	100.000	Y	115.470

<2> Coordinates of P2

(Absolute coordinate)		(Machine coordinate)	
X	200.000	X	257.735
Y	100.000	Y	115.470

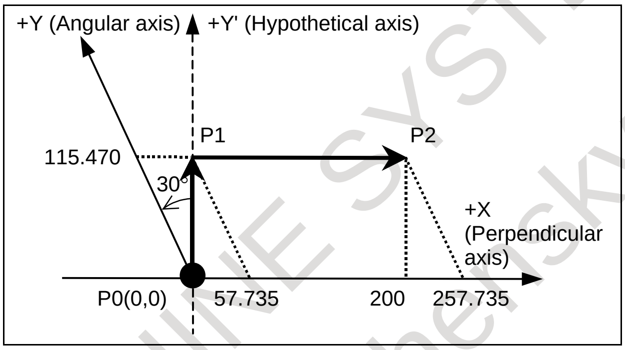


Fig. 1.8 (f)

(2) When the signal for disabling angular axis control for the perpendicular axis (NOZAGC) is set to “1”

<1> Coordinates of P1

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	0.000
Y	100.000	Y	115.470

<2> Coordinates of P2

(Absolute coordinate)		(Machine coordinate)	
X	200.000	X	200.000
Y	100.000	Y	115.470

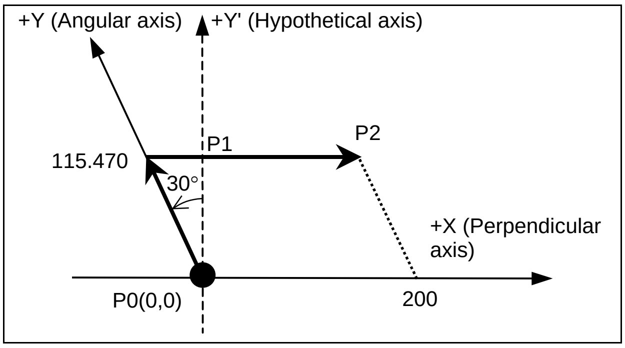


Fig. 1.8 (g)

- 3-dimensional coordinate conversion

In the 3-dimensional coordinate conversion mode, slanted coordinate system conversion is applied to the workpiece coordinate system that has undergone 3-dimensional coordinate conversion.

- Stored stroke limit

Stored stroke limits under angular axis control can be set not in a slanted coordinate system but in the Cartesian coordinate system by setting bits 0, 1, and 2 (AOT, AO2, and AO3) of parameter No. 8201.

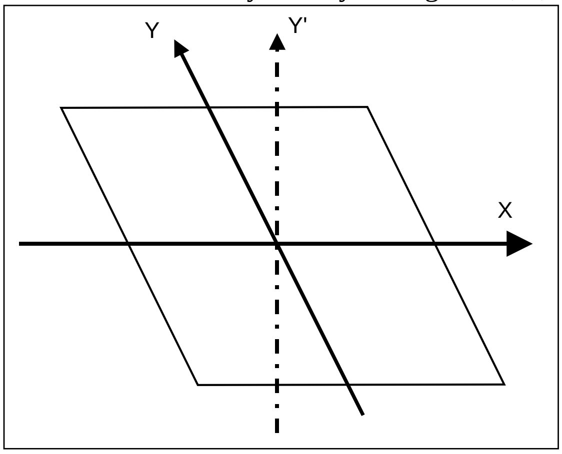


Fig. 1.8 (h) OT area in a slanted coordinate system

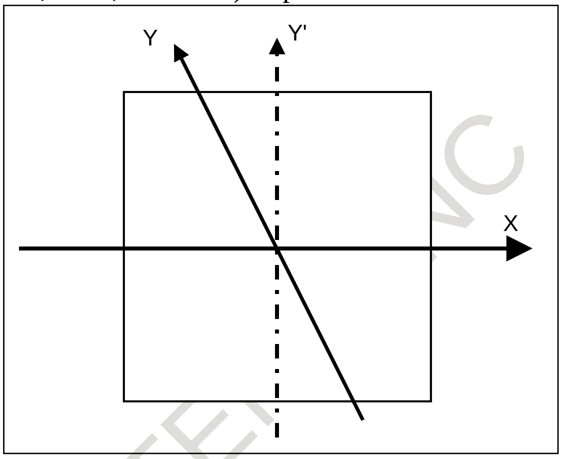


Fig. 1.8 (i) OT area in a Cartesian coordinate system

Machine coordinates include a value converted for the angular axis and a compensation value for the perpendicular axis, so that a slanted machine coordinate system as shown in Fig. 1.8 (h) results.
A stored stroke limit is checked in the machine coordinate system, so that the limit area is slanted to form a rhombus as shown in Fig. 1.8 (h). In this case, the area cannot be identified intuitively. So, stroke limits are checked not in an actual slanted machine coordinate system but in a virtual Cartesian machine coordinate system as shown in Fig. 1.8 (i).

The functions that operate in the Cartesian coordinate system are:
- Stored stroke limit 1 (Both of I and II)
- Stored stroke limit 2 (G22/G23)
- Stored stroke limit 3
- Pre-movement stroke check

The pre-movement stroke check function does not work in a slanted coordinate system. Unless this function is enabled, and the coordinate system is converted to the Cartesian coordinate system, no stroke check is made.
- Stored stroke external setting (function specific to the M series only and valid only for OT1)
- Bit 7 (BFA) of parameter No. 1300 for specifying whether to issue an alarm before or after a stroke limit is exceeded (valid for OT1 and OT3)

The stored stroke limit functions other than the above work in a slanted coordinate system.

- General purpose retract

When the angular axis control is valid the general purpose retract is retracted in the Cartesian coordinate system. Therefore the retracted distance (parameter No.7741) is set with the Cartesian coordinate system. Moreover, even if the command is the machine coordinate system (=the slanted coordinate system) such as G28, G30, G53 the general purpose retract is retracted in the Cartesian coordinate system.

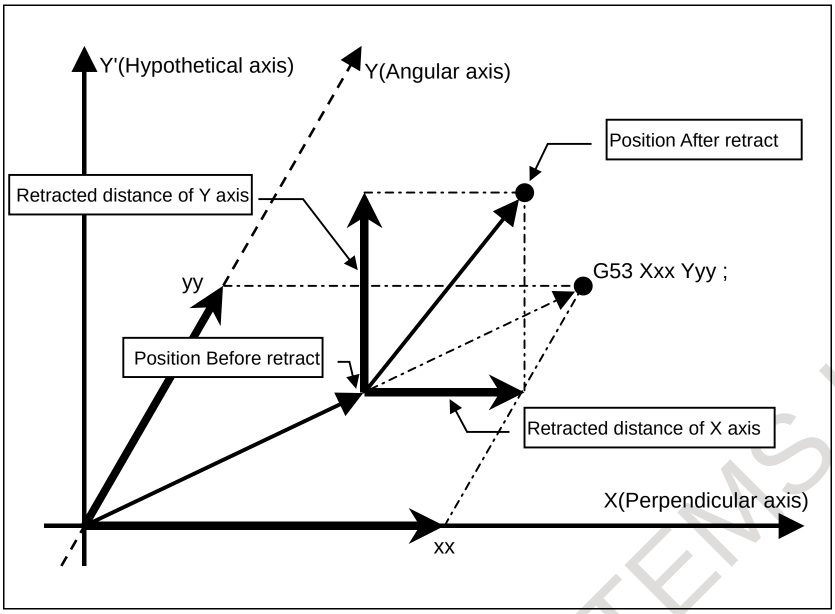


Fig.1.8 (j) Retract motion

- Relationships between this function and axis-by-axis input/output signals

The Table 1.8 (b) and Table 1.8 (c) indicates the relationships between this function and the meaning of each controlled axis signal.

The input/output signals are classified as signals valid for the program coordinate system (Cartesian coordinate system) and signals valid for the machine coordinate system (slanted coordinate system). In the "Classification" column, "Cartesian" is indicated for a signal that is valid for the Cartesian coordinate system, and "Slanted" is indicated for a signal that is valid for the slanted coordinate system.

A signal valid for the Cartesian coordinate system means a signal valid for a specified axis, and a signal valid for the slanted coordinate system is a signal valid for actual machine movement.

This means that when a movement is made along the perpendicular axis by a movement along the angular axis alone:

A signal valid for the Cartesian coordinate system is affected by a movement along the angular axis.

A signal valid for the slanted coordinate system is not affected by a movement along the angular axis.

Table 1.8 (b)

Input signal				
Signal name		**Address**	**Classification**	**Remarks**
Interlock signal for each axis	*ITx	G130	Cartesian	When a movement is made along the angular axis only, interlocking the perpendicular axis does not interlock a movement along the perpendicular axis made by a movement along the angular axis. Caution) When using the axis-by-axis interlock signal, make both of the angular axis and perpendicular axis are set to “1”.
Overtravel	*+Lx *-Lx	G114 G116	Slanted	This signal is applied to each axis independently. (If the perpendicular axis is made ”1”, no alarm is issued for the perpendicular axis even when an OT alarm is issued for the angular axis.)
Deceleration signal for reference position return	*DECx	X009	Slanted	This signal is applied to each axis independently.
Servo-off signal	SVFx	G126	Slanted	This signal is applied to each axis independently.

Input signal				
Signal name		**Address**	**Classification**	**Remarks**
Controlled axis removal signal	DTCHx	G124	Slanted	This signal is applied to each axis independently.
Feed axis direction selection signal	+Jx -Jx	G100 G102	Cartesian	A movement is made in the Cartesian coordinate system. (When the +J/-J signal for the angular axis is made "1", a movement is made also along the perpendicular axis.)
Mirror image	MIx	G106	Slanted	Mirror image is applied to the slanted coordinate system for each axis independently. Caution) Be sure to turn off the mirror image signal for the angular axis and perpendicular axis engaged in manual operation.
Interlock signal for each axis direction, Tool offset write signals	+MIT1, +MIT2	X004.2, 4	Cartesian	Set the tool compensation parameter in the Cartesian coordinate system.
Each-axis machine lock signal	MLKx	G108	Slanted	This signal is applied to each axis independently.

Table 1.8 (c)

Output signal				
Signal name		**Address**	**Classification**	**Remarks**
In-position signal	INPx	F104	Slanted	Applied to each axis independently.
Mirror image check signal	MMIx	F108	Slanted	Applied to each axis independently.
Controlled axis removal in-progress signal	MDTCHx	F110	Slanted	Applied to each axis independently.
Travel in-progress signal	MVx	F102	Slanted	Applied to each axis independently.
Reference position return completion signal	ZPx	F094	Cartesian	Applied to each axis independently. (A manual reference position return operation and the first automatic reference position return operation after power-up need to be performed first for the angular axis.)
2nd reference position return completion signal	ZP2x	F096	Cartesian	Applied to each axis independently.
3rd reference position return completion signal	ZP3x	F098	Cartesian	Applied to each axis independently.
4th reference position return completion signal	ZP4x	F100	Cartesian	Applied to each axis independently.

- Pitch Error Compensation

The pitch error compensation is applied in a slanted coordinate system.

Limitation

- 3-dimensional coordinate conversion

If the basic three axes in the 3-dimensional coordinate conversion mode do no include a perpendicular axis and angular axis for angular axis control, operation cannot be performed normally in a correct slanted coordinate system.

- Linear scale with an absolute address reference mark

- For both of the angular axis and perpendicular axis, a linear scale with an absolute address reference mark must be used.
- Reference position return operation must be first completed along the angular axis.

- Return operation cannot be performed along the perpendicular axis while return operation is being performed along the angular axis.

- Synchronous control

- To perform synchronous control over related axes under angular axis control, specify the angular axis and Cartesian axis on the master axis side and the angular axis and Cartesian axis on the slave axis side as the targets of synchronous control at the same time. In synchronous control, one angular axis must be paired with the other angular axis while one Cartesian axis must be paired with the other Cartesian axis.
  If synchronous control is performed in a way other than the above, alarm PS0375 is issued.
  Example)

Path 1		Path 2
X1 (Cartesian axis)	← Synchronous control pair	→ X2 (Cartesian axis)
Y1 (slanted axis)	← Synchronous control pair	→ Y2 (slanted axis)

- Composite control

- To perform composite control over related axes under angular axis control, specify the angular axis and Cartesian axis on the master axis side and the angular axis and Cartesian axis on the slave axis side as the targets of composite control at the same time. In composite control, one angular axis must be paired with the other angular axis while one Cartesian axis must be paired with the other Cartesian axis.
  If composite control is performed in a way other than the above, alarm PS0375 occurs.
  Example)

Path 1		Path 2
X1 (Cartesian axis)	← Composite control pair	→ X2 (Cartesian axis)
Y1 (slanted axis)	← Composite control pair	→ Y2 (slanted axis)

- Rigid tapping

- An angular axis cannot be used as a tapping axis for rigid tapping.

- Manual handle interrupt

- When machine coordinate system setting (G53) or return to reference position (G28, G30) is commanded to the perpendicular axis, the manual handle interrupt can not be used to the angular axis.

- Functions that cannot be used simultaneously

- Axis synchronous control, Hypothetical axis control, EGB function, PMC axis control, Superimposed control, Flexible synchronization Control, Flexible path axis assignment, High precision oscillation function, Reference position setting with mechanical stopper, Reference position setting with mechanical stopper by grid method

Caution

⚠ CAUTION

1 After angular axis control parameter setting, be sure to perform manual reference position return operation.
2 Before manual reference position return operation is performed along the perpendicular axis, reference position return operation along the angular axis must be completed (with the reference position return completion signal for the angular axis (ZPx) set to 1). If reference position return operation is performed along the perpendicular axis first, an alarm (PS0372) is issued.
3 When the setting is made so that the tool moves along the perpendicular axis during manual reference position return along the slanted axis (bit 2 (AZK) of parameter No. 8200 is set to 0), if once manual reference position return has been performed along the angular axis, also perform manual reference position return along the perpendicular axis immediately after the operation.
4 Before attempting to manually move the tool along the angular and perpendicular axes independently, set signal for disabling angular axis control for the perpendicular axis NOZAGC to “1”.
5 Once the tool has been moved along the angular axis when signal for disabling angular axis control for the perpendicular axis NOZAGC has been set to “1”, manual reference position return must be performed.
6 The same increment system must be used with the angular axis and perpendicular axis.
7 Before a perpendicular axis reference position return check can be made, angular axis reference position return operation must be completed.
8 No rotary axis must be set for the angular axis and perpendicular axis. A rotary axis may be specified only for a linear axis.
9 Set a position switch operation range (parameter Nos. 6930 to 6965) in a slanted coordinate system.

Signal

Signal for disabling angular axis control for the perpendicular axis NOZAGC<Gn063.5>

[Classification] Input signal
[Function] Disables angular axis control for the perpendicular axis.
[Operation] When this signal is set to “1”, the control unit operates as follows:
- Converts an angular axis move command to angular coordinates. The perpendicular axis is not affected by an angular axis move command, however.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8200					AZP	AZR		AAC

[Input type] Parameter input
[Data type] Bit path

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 AAC

0: Does not perform angular axis control.
1: Performs inclined axis control.

#2 AZR

0: The machine tool is moved along the Cartesian axis during manual reference position return along the slanted axis under angular axis control.
1: The machine tool is not moved along the Cartesian axis during manual reference position return along the slanted axis under angular axis control.

#3 AZP When a movement is made along the Cartesian axis due to a movement along the slanted axis, reference position return end signals for the Cartesian axis ZP1 to ZP8 <Fn094.0 to Fn094.7> are:
0: Not cleared.
1: Cleared.

	#7	#6	#5	#4	#3	#2	#1	#0
8201	ADG	A53				AO3	AO2	AOT

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 AOT Stored stroke limit 1 under angular axis control is handled as:
0: Value in the slanted coordinate system.
1: Value in the Cartesian coordinate system.

#1 AO2 Stored stroke limit 2 under angular axis control is handled as:
0: Value in the slanted coordinate system.
1: Value in the Cartesian coordinate system.

#2 AO3 Stored stroke limit 3 under angular axis control is handled as:
0: Value in the slanted coordinate system.
1: Value in the Cartesian coordinate system.

#6 A53 So far, if a slanted axis is singly specified by a machine coordinate command (G53) in angular axis control, this parameter set to 0 specifies that "compensation is applied to the Cartesian axis", and this parameter set to 1 specifies that "a movement is made along the slanted axis only". However, the specification has been changed so that "a movement is made along the slanted axis only", regardless of whether this parameter is set to 0 or 1.

#7 ADG The contents of diagnostic data Nos. 306 and 307 are:
0: Not swapped. The slanted axis and Cartesian axis are displayed in this order.
1: Swapped. The Cartesian axis and slanted axis are displayed in this order.

	#7	#6	#5	#4	#3	#2	#1	#0
8209								ARF

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter bit is set, the power must be turned off before operation is continued.

#0 ARF In angular axis control, a movement from an intermediate point to the reference position in the G28/G30 command is:
0: Made in the angular coordinate system.
1: Made in the Cartesian coordinate system.

8210	Slant angle of a slanted axis in angular axis control

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] -180.000 to 180.000. However, angular axis control is disabled in the ranges -95.000 to -85.000 and 85.000 to 95.000 (in the case of IS-B).

8211	Axis number of a slanted axis subject to angular axis control
8212	Axis number of a Cartesian axis subject to slanted axis control

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to number of controlled axes
When angular axis control is to be applied to an arbitrary axis, these parameters set the axis numbers of a slanted axis and Cartesian axis. If 0 is set in either of the two parameters, the same number is set in the two parameters, or a number other than the controlled axis numbers is set in either of the two parameters, a slanted axis and Cartesian axis are selected as indicated in the following table:

	Slanted axis	Cartesian axis
M series	Y-axis (axis with 2 set in parameter No. 1022) of the basic three axes	Z-axis (axis with 3 set in parameter No. 1022) of the basic three axes
T series	X-axis (axis with 1 set in parameter No. 1022) of the basic three axes	Z-axis (axis with 3 set in parameter No. 1022) of the basic three axes

Alarm and message

Number	Message	Description
PS0372	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the orthogonal axis before the completion of a return to the reference position on the angular axis. However, this attempt failed because a manual return to the reference position during angular axis control or an automatic return to the reference position after power-up was not commanded. First, return to the reference position on the angular axis, then return to the reference position on the orthogonal axis.

Number	Message	Description
PS0375	CAN NOT ANGULAR CONTROL(SYNC:MIX:OVL)	Angular axis control is disabled for this axis configuration. 1) When some related axes under angular axis control are not in synchronous control mode or when one angular axis is not paired with the other angular axis or one Cartesian axis is not paired with the other Cartesian axis in synchronous control 2) When some related axes under composite control are not in composite control mode or when one angular axis is not paired with the other angular axis or one Cartesian axis is not paired with the other Cartesian axis in composite control 3) When related axes under angular axis control is switched to superimposed control mode1)

Diagnosis

306	Machine coordinates on the angular axis in the Cartesian coordinate system
307	Machine coordinates on the perpendicular axis in the Cartesian coordinate system

[Data type] Real number
[Unit of data] Machine unit

Machine coordinates in the Cartesian coordinate system are displayed in angular axis control.
Bit 7 (ADG) of parameter No. 8201 can be used to change the display order.

1.9 ELECTRONIC GEAR BOX

1.9.1 Electronic Gear Box

Overview

This function enables fabrication of high-precision gears, screws, and other components by rotating the workpiece in synchronization with a rotating tool or by moving the tool in synchronization with a rotating workpiece. The rate of synchronization can be specified with a program. The synchronization of tool and workpiece axes with this function adopts a system in which the synchronization is directly controlled by digital servo, so that the workpiece axis can follow up the speed fluctuations on the tool axis with no error, thereby allowing fabrication of high-precision cogwheels. In the subsequent explanation, the Electronic Gear Box is called the EGB.

- Example of controlled axis configuration

Spindle : EGB master axis : Tool axis
1st axis : X axis
2nd axis : Y axis
3rd axis : C axis (EGB slave axis : Workpiece axis)
4th axis : C axis (EGB dummy axis : Cannot be used as a normal controlled axis.)

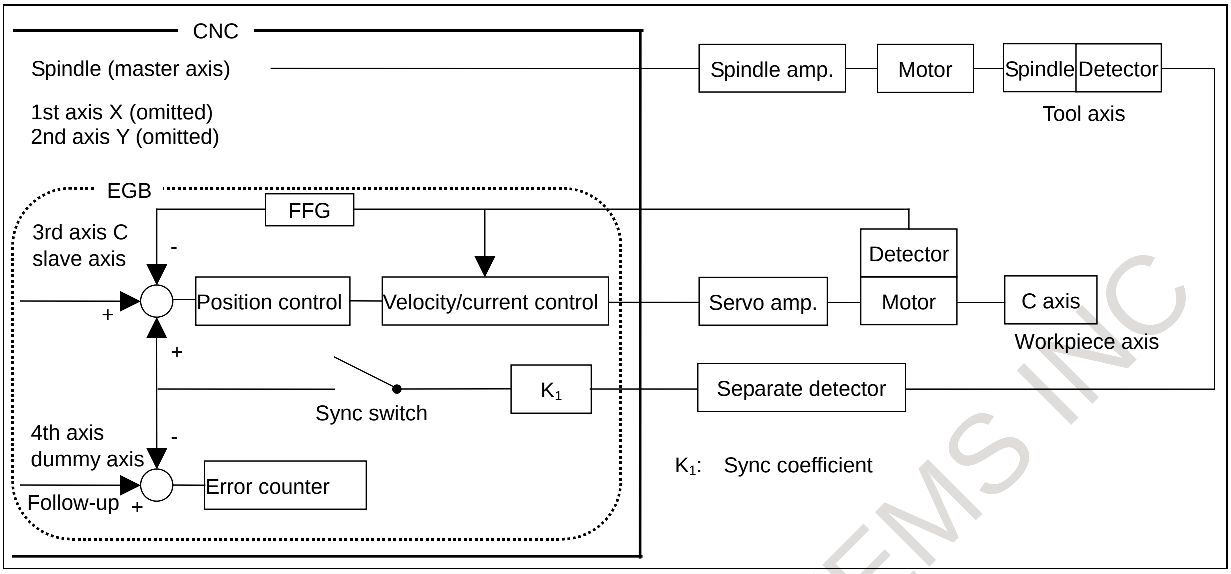


Fig. 1.9.1 (a)

For EGB axis configuration parameter setting examples, see the section on "FSSB setting".

Format

	Bit 0 (EFX) of parameter No.7731=0	Bit 0 (EFX) of parameter No.7731=1	
		Bit 5 (HBR) of parameter No.7731=1	Bit 5 (HBR) of parameter No.7731=0
Start of synchronization	**G81 T_ (L_) (Q_ P_) ;**	**G81.4 R_ (L_) (Q_ P_) ;**	**G81.4 T_ (L_) (Q_ P_) ;**
Cancellation of synchronization	**G80 ;**	**G80.4 ;**	**G80.4 ;**
	(*1) (*4)	**(*2) (*4)**	**(*3) (*4)**

T(or R) : Number of teeth (Specifiable range: 1 to 5000)
L : Number of hob threads (Specifiable range: -250 to 250)
The sign of L determines the direction of rotation for the workpiece axis.
When L is positive, the direction of rotation for the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation for the workpiece axis is negative (- direction).
When L is 0, it follows the setting of bit 3 (LZR) of parameter No.7701.
If L is not specified, the number of hob threads is assumed 1.
Q : Module or diametral pitch
Specify a module in the case of metric input.
(Unit: 0.00001mm, Specifiable range: 0.01 to 25.0mm)
Specify a diametral pitch in the case of inch input.
(Unit: 0.00001inch^{-1}, Specifiable range: 0.01 to 254.0 inch^{-1})
P : Gear helix angle
(Unit: 0.0001deg, Specifiable range: -90.0 to 90.0deg)
*1 Use it for machining centers.
*2 Use it for lathes.
*3 Use it for machining centers.
This format enables specification of the same G codes as those for lathes.
*4 When specifying Q and P, the user can use a decimal point.

NOTE
Specify G81, G80, G81.4, and G80.4 in a single block.

Explanation

- Master axis, slave axis, and dummy axis

The synchronization reference axis is called the master axis, while the axis along which movement is performed in synchronization with the master axis is called the slave axis. For example, if the workpiece moves in synchronization with the rotating tool as in a hobbing machine, the tool axis is the master axis and the workpiece axis is the slave axis.
Which axes to become the master and slave axes depends on the configuration of the machine. For details, refer to the manual issued by the machine tool builder.
A single servo axis is used exclusively so that digital servo can directly read the rotation position of the master axis. (This axis is called the EGB dummy axis.)

- Synchronous control

(1) Start of synchronization

If G81 is issued so that the machine enters synchronization mode, the synch switch of the EGB function is closed, and the synchronization of the tool and workpiece axes is started. At this time, the EGB mode signal SYNMOD <Fn065.6> becomes “1”. During synchronization, the rotation about the tool and workpiece axes is controlled so that the relationship between T (number of teeth) and L (number of hob threads) is maintained. During synchronization, the synchronization relationship is maintained regardless of whether the operation is automatic or manual.
Specify P and Q to use helical gear compensation.
If only either P or Q is issued, alarm PS1594, “EGB FORMAT ERROR” is generated.
If, during synchronization, G81 is issued again without synchronization cancellation, alarm PS1595, “ILL-COMMAND IN EGB MODE” is generated if bit 3 (ECN) of parameter No. 7731 is 0. If bit 3 (ECN) of parameter No. 7731 is 1, helical gear compensation is conducted with the synchronization coefficient being changed to the one newly specified with T and L commands if T and L commands are issued, and if T and L commands are not issued and only P and Q commands are issued, helical gear compensation is conducted with the synchronization coefficient kept intact. This allows consecutive fabrication of helical gears and spur gears.

(2) Start of tool axis rotation

When the rotation of the tool axis starts, the rotation of the workpiece axis starts so that the synchronous relationship specified in the G81 block can be maintained.
The rotation direction of the workpiece axis depends on the rotation direction of the tool axis. That is, when the rotation direction of the tool axis is positive, the rotation direction of the workpiece axis is also positive; when the rotation direction of the tool axis is negative, the rotation direction of the workpiece axis is also negative. However, by specifying a negative value for L, the rotation direction of the workpiece axis can be made opposite to the rotation direction of the tool axis.
During synchronization, the machine coordinates of the workpiece axis and EGB axis are updated as synchronous motion proceeds. On the other hand, a synchronous move command has no effect on the absolute and relative coordinates.

(3) Termination of tool axis rotation

In synchronism with gradual stop of the tool axis, the workpiece axis is decelerated and stopped. By specifying the G80 command after the spindle stops, synchronization is canceled, and the EGB synchronization switch is opened. At this time, the EGB mode signal SYNMOD becomes “0”.

(4) Cancellation of synchronization

When cancellation of synchronization is issued, the absolute coordinate on the workpiece axis is updated in accordance with the amount of travel during synchronization. Subsequently, absolute commands for the workpiece axis will be enabled.
For a rotation axis, the amount of travel during synchronization, as rounded to 360-degree units is added to the absolute coordinate.
In the G80 block, only O and N addresses can be specified.
By setting bit 0 (HBR) of parameter No. 7700 to 0, it is possible to cancel synchronization with a reset.
Synchronization is automatically canceled under the following conditions:

<1> An emergency stop is applied.
<2> A servo alarm is generated.
<3> Alarm PW0000, “POWER MUST BE OFF” is generated.
<4> An IO alarm is generated.

⚠ CAUTION

1 Feed hold, interlock, and machine lock are invalid to a slave axis in EGB synchronization.
2 Even if an OT alarm is issued for a slave axis in EGB synchronization, synchronization will not be canceled.
3 During synchronization, it is possible to execute a move command for a slave axis and other axes, using a program. The move command for a slave command must be an incremental one.

NOTE

1 If bit 0 (HBR) of parameter No. 7700 is set to 1, EGB synchronization will not be canceled due to a reset. Usually, set this parameter bit to 1.
2 In synchronization mode, it is not possible to specify G27, G28, G29, G30, and G53 for a slave axis.
3 It is not possible to use controlled axis detach for a slave axis.
4 During synchronization, manual handle interruption can be performed on the slave and other axes.
5 In synchronization mode, no inch/metric conversion commands (G20 and G21) cannot be issued.
6 In synchronization mode, only the machine coordinates on a slave axis are updated.
7 If bit 0 (EFX) of parameter No. 7731 is 0, no canned cycle for drilling can be used. To use a canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.
8 If bit 0 (TDP) of parameter No. 7702 is 1, the permissible range of T is 0.1 to 500 (1/10 of the specified value).
9 If, at the start of EGB synchronization (G81), L is specified as 0, synchronization starts with L assumed to be 1 if bit 3 (LZR) of parameter No.7701 is 0; if bit 3 (LZR) of parameter No.7701 is 1, synchronization is not started with L assumed to be 0. At this time, helical gear compensation is performed.
10 Feed per revolution is performed on the feedback pulses on the spindle. By setting bit 0 (ERV) of parameter No. 7703 to 1, feed per revolution can be performed based on the speed on the synchronous slave axis.
11 Actual cutting feedrate display does not take synchronization pulses into consideration.
12 For an EGB slave axis, synchronous and composite control cannot be executed.
13 In EGB synchronization mode, AI contour control mode is temporarily canceled.
14 Not advanced preview feed-forward but conventional feed-forward is enabled in the path where EGB synchronization mode is effective.

- Synchronization start/cancellation timing chart example

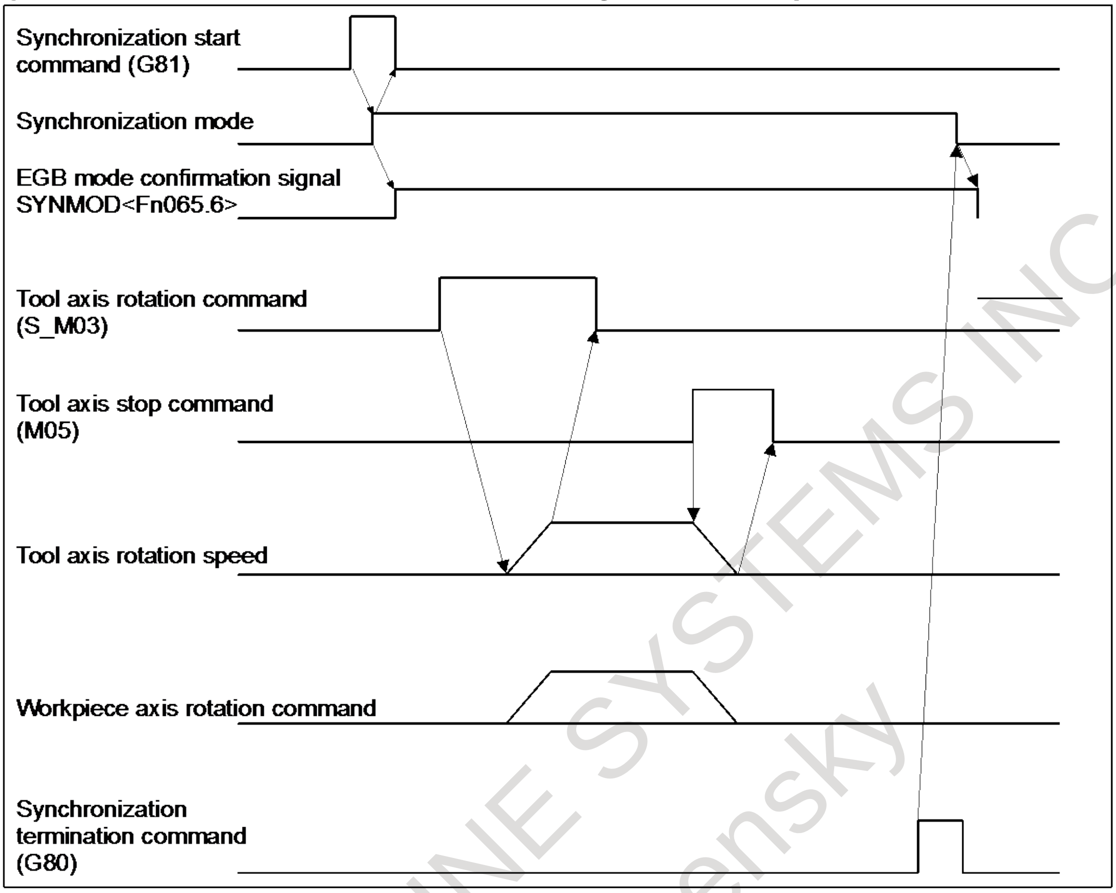


Fig. 1.9.1 (b)

- How to reduce the synchronous error

When you use the Electronic gear box function, to reduce the synchronous error, please apply feed-forward to the slave axis and set 100% to the parameter of feed-forward coefficient.
And please confirm the effectiveness of feed-forward by the following procedure.

[Procedure]

1. When the slave axis synchronizes only with the command from master axis (i.e. When the slave axis doesn't use helical gear compensation), the position error of slave axis is regarded as the synchronous error. Please check that the position error (DGN data No.300) of the slave axis becomes 0 or so.
2. And also please check the position error is near 0 even when the speed of the master axis is changed.

Please set the following parameters to use Feed-forward function with 100% coefficient.

[Setting parameters]

Bit 3 (PIEN) of parameter No. 2003 = 1 (Slave axis)	Use PI control in velocity control
Bit 1 (FEED) of parameter No. 2005 = 1 (Slave axis)	Use Feed-forward function
Bit 1 (FFALx) of parameter No. 2011 = 1 (Slave axis)	Use Feed-forward function irrespective of feed mode
Parameter No.2068 (FF coefficient) = 10000 (Slave axis)	Feed-forward coefficient is 100%.

Please refer to the chapter of “Feed-forward Function” in FANUC AC SERVO MOTOR α*i* series FANUC AC SERVO MOTOR β*i* series FANUC LINEAR MOTOR L*i*S series FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR D*i*S series Parameter manual (B-65270EN) about the detail of Feed-forward function.

- How to reduce shock at the start of acc./dec.

If the shock of slave axis is large when the master axis accelerates or decelerates in velocity control mode, please apply “Soft start/stop” function to the master axis (spindle axis).
Please set the following parameters to use Soft start/stop function.

[Setting parameters]

Bit 2 (SOSALWs) of parameter No. 4399 = 1 Use Soft start/stop function even at emergency stop

Note) If the spindle axis is a sub axis of spindle switching control, please set bit 2 of parameter No. 4472 instead of bit 2 of parameter No. 4399.

Parameter No. 4030 Soft start/stop setting time
Parameter No. 4508 Rate of change in acceleration at soft start/stop

Note) Parameters Nos. 4030 and 4508 should be tuned according to the spindle characteristic to reduce the shock well.

[Signals]

Soft start/stop signal SOCNA <Gn071.4> : First spindle
Soft start/stop signal SOCNB <Gn075.4> : Second spindle
Soft start/stop signal SOCNC <Gn205.4> : Third spindle
Soft start/stop signal SOCND <Gn267.4> : Fourth spindle

Please refer to FANUC AC SPINDLE MOTOR α*i*/β*i* series BUILT-IN SPINDLE MOTOR B*i* series parameter manual (B-65280EN) about the detail of Soft start/stop function.

- Helical gear compensation

For a helical gear, the workpiece axis is compensated for the movement along the Z-axis (axial feed axis) based on the torsion angle of the gear.
Helical gear compensation is performed with the following formulas:

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (for metric input)}$$

$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (for inch input)}$$

where

Compensation angle: Signed absolute value (deg)

Z : Amount of travel on the Z-axis after the specification of G81
P : Signed gear helix angle (deg)
π : Circular constant
T : Number of teeth
Q : Module (mm) or diametral pitch (inch^{-1})

Use P, T, and Q specified in the G81 block.

In helical gear compensation, the machine coordinates on the workpiece axis and the absolute coordinates are updated with helical gear compensation.
The updated timing is as follows.

<table>
<tr><th>Machine coordinates</th><th colspan="2">Absolute coordinates</th></tr>
<tr><td rowspan="3">During helical gear compensation</td><td colspan="2">Bit 7(HAD) of parameter No.7731</td></tr>
<tr><td>= 0</td><td>= 1</td></tr>
<tr><td>Synchronization cancel</td><td>During helical gear compensation</td></tr>
</table>

- Direction of helical gear compensation

The direction depends on bit 2 (HDR) of parameter No. 7700.

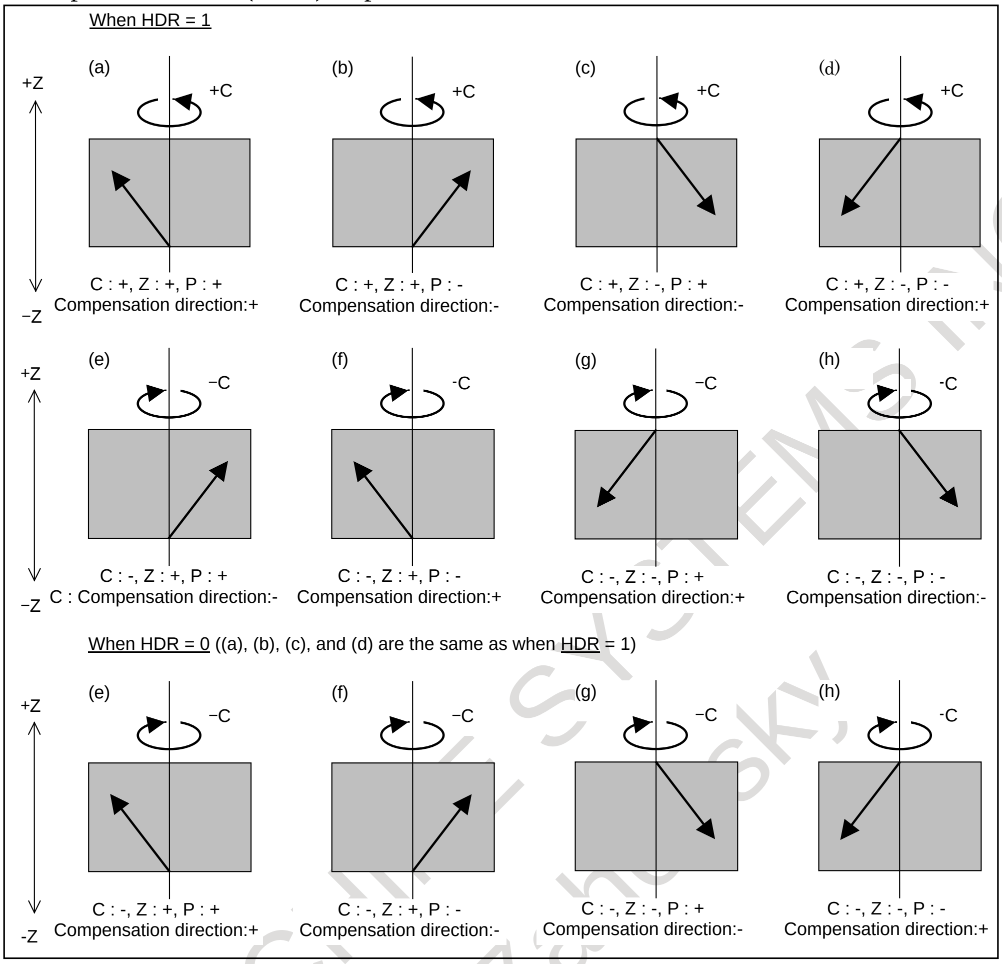


Fig. 1.9.1 (c) Direction of helical gear compensation

- Synchronization coefficient

A synchronization coefficient is internally represented using a fraction (Kn/Kd) to eliminate an error. The formula below is used for calculation.

$$\text{Synchronization coefficient} = \frac{K_n}{K_d} = \frac{L}{T} \times \frac{\beta}{\alpha}$$

where

L : Number of hob threads

T : Number of teeth

α : Number of pulses of the position detector per rotation about the master axis (parameter No. 7772)

β : Number of pulses of the position detector per rotation about the slave axis (parameter No. 7773)

Kn / Kd is a value resulting from reducing the right side of the above formula, but the result of reduction is subject to the following restrictions:

$-2147483648 \le K_n \le 2147483647$

$1 \le K_d \le 2147483647$

When this restriction is not satisfied, the alarm PS1596, “EGB OVERFLOW” is issued when G81 is specified.

- Retract function

(1) Retract function with an external signal

When the retract signal, RTRCT<Gn066.4>, becomes “1” (the rise of the signal is captured), retraction is performed with the retract amount set in parameter No. 7741 and the speed set in parameter No. 7740.
No movement is performed along an axis for which 0 is set as the retract amount.
After the end of retraction, the retract completion signal, RTRCTF<Fn065.4>, is output.

(2) Retract function with an alarm

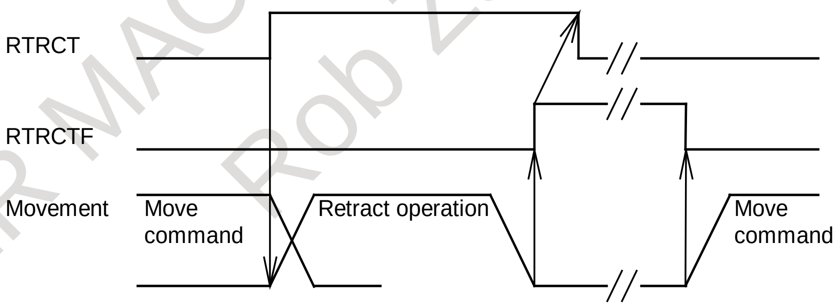
If, during EGB synchronization or automatic operation, a CNC alarm is issued, retraction is performed with the retract amount set in parameter No. 7741 and the speed set in parameter No. 7740.
This can prevent the tool and the object being machined from damage if a servo alarm is generated.
No movement is performed along an axis for which 0 is set as the retract amount.
After the end of retraction, the retract completion signal, RTRCTF, is output.

Conditions under the retract function with an alarm
The conditions under which the retract function with an alarm can be changed using the settings of bit 1 (ARE) of parameter No. 7703 and bit 2 (ARO) of parameter No. 7703.
The table below lists parameter settings and corresponding conditions.

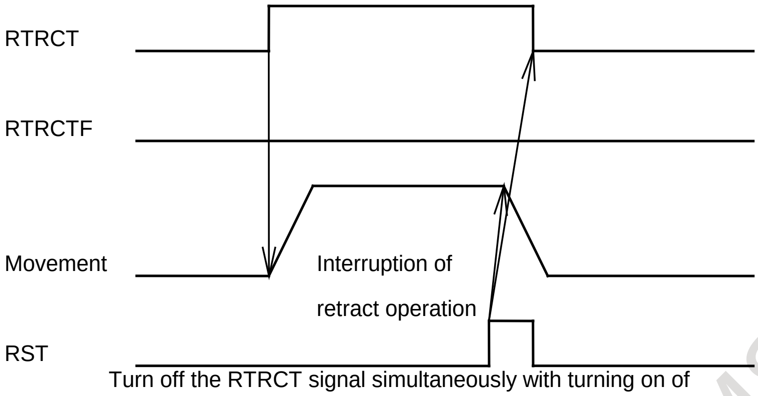
ARE	ARO	Condition
1	0	EGB synchronization is in progress.
1	1	Both EGB synchronization and automatic operation are in progress.
0	0	Either EGB synchronization or automatic operation is in progress.
0	1	

Timing chart

<1> On/off timing of RTRCT and RTRCTF

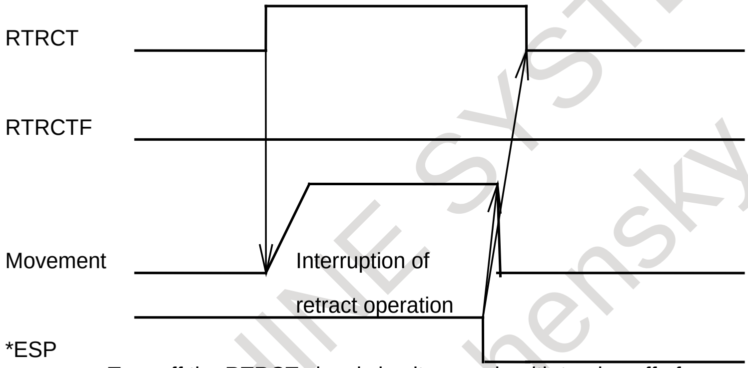


<2> Interruption of retraction with to a reset



Turn off the RTRCT signal simultaneously with turning on of the RST signal.

<3> Interruption of retraction due to an emergency stop



Turn off the RTRCT signal simultaneously with turning off of the *ESP signal.

> ⚠ **CAUTION**
> 1 Retraction is performed at the speed specified in parameter No. 7740.
> 2 Feed hold is not effective to movement during retraction.
> 3 Feedrate override is not effective to movement during retraction.

NOTE

1 During a retract operation, an interlock is effective to the retract axis.
2 During a retract operation, a machine lock is effective to the retract axis. The retract operation terminates in the machine lock state, and a retract completion signal is output.
3 The retraction direction depends on the movement direction of the machine, regardless of whether an mirror image (signal and setting) is enabled or disabled. (No mirror image can be applied to the updating of absolute coordinates.)
4 If retraction is performed during automatic operation, automatic operation is halted simultaneously with a retract operation, but it is at the end of the retract operation that the operation state switches to the automatic operation halt state.

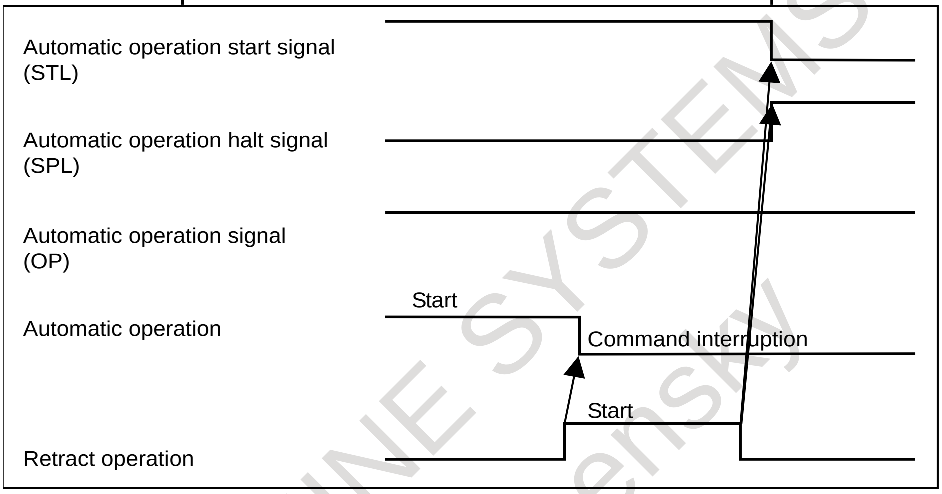


Fig. 1.9.1 (d)

5 It is not possible to perform automatic operation during retraction.
6 The acceleration/deceleration of a retract operation is in the acceleration/deceleration state at the start of retraction.
7 Retract movement is performed with non-linear type positioning.
8 If, during a retract operation, a reset or an emergency stop is made, the operation is interrupted. At this time, the retract completion signal does not become “1”.
9 To enable the retract function with an alarm, bit 3 (ART) of parameter No.7702 must be set.
10 The retract function with an alarm does not perform a retract operation on the retract axis if an overtravel alarm or a servo alarm is generated on the retract axis.
11 If a new alarm is issued during retraction with the retract function with an alarm, a retract operation is not performed.
12 After the end of retraction with a servo alarm, servo position control stops in 400 ms.
13 The retract completion signal RTRCTF <Fn065.4> becomes “0” when a move command is issued for any axis after retract operations on all axes are completed.

Example

O1000 ;	
N0010 M19 ;	Tool axis orientation
N0020 G28 G91 C0 ;	Reference position return on the work axis
N0030 G81 T20 L1 ;	Start of synchronization of the tool axis and the work axis (Rotation about the work axis by 18° per rotation about the tool axis)
N0040 S300 M03 ;	Rotation about the tool axis at 300min^{-1}
N0050 G01 X_ F_ ;	Movement along the X-axis (cutting)
N0060 G01 Z_ F_ ;	Movement along the Z-axis (machining)
;	If required, C, X, Z, and other axis command are possible.
;	
N0100 G01 X_ F_ ;	Movement along the X-axis (escape)
N0110 M05 ;	Stop on the tool axis
N0120 G80 ;	Cancellation of synchronization of the tool and work axes
N0130 M30 ;	

Signal

Retract signal RTRCT <Gn066.4>

[Classification] Input signal

[Function] Performs retraction for the axis specified with a parameter.

[Operation] When this signal becomes “1”, the CNC operates as follows:

The CNC can capture the rise of the signal and perform retraction for the axis for which the retract amount has been specified with parameter No. 7741. The retract amount and the retract speed will be the values previously set in parameters Nos. 7741 and 7740. After the end of retraction, the retract completion signal, RTRCTF, is output. The retract signal is effective both during automatic operation mode (MEM, MDI, etc.) and manual operation mode (HNDL, JOG, etc.). If, during automatic operation, the retract signal is set to “1”, a retract operation is performed and, at the same time, automatic operation is halted.

Retract completion signal RTRCTF <Fn065.4>

[Classification] Output signal

[Function] Posts notification of the completion of retraction.

[Operation] This signal is set to “1” in the following case:

- Upon the completion of retraction (at the end of movement)

This signal is set to “0” in the following case:

- When a move command is issued for any retract axis after the end of a retract operation.

> **NOTE**
> The retract signal is not accepted while the retract completion signal is set to “1”.

EGB mode signal SYNMOD <Fn065.6>

[Classification] Output signal

[Function] Posts notification that synchronization using the EGB is in progress.

[Operation] This signal is set to “1” in the following case:

- While synchronization using the EGB is in progress

This signal is set to “0” in the following case:

- Once synchronization using the EGB has terminated

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn066				RTRCT				

	#7	#6	#5	#4	#3	#2	#1	#0
Fn065		SYNMOD		RTRCTF				

Parameter

The table below gives parameters related to EGB.

Parameter number	Description
1006#0 1006#1	An EGB slave axis and an EGB dummy axis require that the setting of a rotary axis (type A) (bit 0 (ROT) of parameter No. 1006 be 1 and bit 1 (ROS) of parameter No. 1006 be 0.
1023	Set from the FSSB setting screen. For FSSB manual setting, be sure to set the EGB axis as described below: The slave axis must be set with an odd number, and the dummy axis with an even number. They must be consecutive. Example: If the servo axis number of the slave axis is 1, the servo axis number of the dummy axis must be set to 2. If the servo axis number of the slave axis is 3, the servo axis number of the dummy axis must be set to 4.
2011#0	Specify an axis to be synchronized. Specify 1 for both an EGB slave axis and EGB dummy axis.
3115#0	The current position is not indicated for an axis for which this parameter is set to 1.Since the current position for an EGB dummy axis has no meaning, set this parameter to 1 to delete the current position indication for the axis from the screen.
7700#0	The synchronization mode is canceled (0)/not canceled (1) by a reset.
7700#2	Compensation direction for helical gear compensation
7701#3	At the start of synchronization (G81), synchronization is started (0)/not started (1) if the number of hob threads L is specified as 0.
7702#0	The specifiable number of teeth, T, at the start of synchronization (G81) is not reduced to a 1/10 of a specified value (0)/reduced (1).
7702#3	The retract function with an alarm is disabled (0)/enabled (1).
7703#0	During synchronization (G81), feed per revolution is performed for feedback pulses (0)/pulses converted to the speed for the workpiece axis(1).
7703#1,#2	Specify when to perform a retract operation with the retract function with an alarm; during synchronization; during synchronization and automatic operation; or during synchronization or automatic operation.
7709	Number of the axial feed axis in helical gear compensation
7731#0	The EGB command is G80 and G81(0)/G80.4 and G81.4(1).
7731#3	When the automatic phase synchronization function for the electronic gear box is disabled, the G81 command cannot be issued again (an alarm is issued) (0)/can be issued again (1)during EGB synchronization.
7731#5	In EGB synchronization start command G81.4, the number of teeth is specified in T (0)/specified in R (1).
7740	Feedrate during retraction
7741	Retract amount
7772	Number of position detector pulses per rotation about tool axis
7773	Number of position detector pulses per rotation about workpiece axis

For FSSB settings, see the section on "FSSB settings".

If FSSB setting mode is automatic setting mode, setting is made automatically by inputting data to the FSSB setting screen. For the slave/dummy axes of EGB, set the value in the 'M/S' item in the FSSB axis setting screen same way of the tandem setting.

Note the following points when specifying parameters for the electronic gear box.

1 Specify an axis that is not used or the same name as that for a slave axis for the name of a dummy axis. Do not use a name which is usually not allowed to be used as an axis address, such as D.

2 Specify the same values for an EGB slave axis and an EGB dummy axis in the following parameters.

1013#0 to 3	Increment system
1004#7	Ten times minimum input increment
1006#0,1	Rotary axis setting
1006#3	Diameter/radius specification
1420	Rapid traverse rate
1421	Rapid-traverse override F0 speed
1820	Command multiplication
2000 and over, except 2022, 2084, 2085	Parameters related to digital servo

3 Specify the amount of travel per rotation about a rotation axis for a slave axis and dummy axis in a parameter No. 1260.

4 Make the specification for a dummy axis in the following way.

1815#1	Whether to use separate detectors. Although an EGB dummy axis uses the interface of a separate detector, set these parameters to 0.
2022	Motor rotation direction. Set the parameter to 111.
2084	Numerator of flexible feed gear ratio. Set the parameter to 1.
2085	Denominator of flexible feed gear ratio. Set the parameter to 1. When using a serial type position detector, set an appropriate value for the flexible feed gear. To set the Flexible Feed Gear to a value other than 1/1 with A/B phase type position detector, set parameter EGFx (bit6 of parameter No.2273) to 1.

5 Reducing synchronous errors requires enabling the feed-forward function for the slave axis. For details, see “How to reduce the synchronous error” in "Explanation" of this chapter.

6 Reducing shocks that may occur at the beginning of acceleration/deceleration requires enabling the soft start/stop function for the spindle axis. For details, see “How to reduce shock at the start of acc./dec.” in "Explanation" of this chapter.

1023	Number of the servo axis for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 80

This parameter associates each control axis with a specific servo axis. Specify values 1+8n, 2+8n, 3+8n, 4+8n, 5+8n, and 6+8n (n = 0, 1, 2, …, 9) like 1, 2, 3, 4, 5, …, 77, and 78.

The control axis number is the order number that is used for setting the axis-type parameters or axis-type machine signals

* For electronic gear box (EGB) controlled axes, two axes need to be specified as one pair. So, make a setting as described below.
EGB axis:
For a slave axis, set an odd (1, 3, 5, 7, 9, ...) servo axis number. For a dummy axis to be paired, set a value obtained by adding 1 to the value set for the slave axis.

	#7	#6	#5	#4	#3	#2	#1	#0
2011								SYNx

[Input type] Parameter input
[Data type] Bit axis

#0 SYNx When the electronic gear box function (EGB) is used, this bit sets the axis to be synchronized.

0: Axis not synchronized by EGB
1: Axis synchronized by EGB
Set 1 for both of the slave and dummy axes of EGB.

> **NOTE**
> The setting of this parameter becomes valid after the power is turned off then back on.

	#7	#6	#5	#4	#3	#2	#1	#0
3115								NDPx

[Input type] Parameter input
[Data type] Bit axis

#0 NDPx The current position is:
0: Displayed.
1: Not displayed.

> **NOTE**
> When using the electronic gear box (EGB) function, set 1 for the EGB dummy axis to disable current position display.

	#7	#6	#5	#4	#3	#2	#1	#0
7700						HDR		HBR

[Input type] Parameter input
[Data type] Bit path

#0 HBR When the electronic gear box (EGB) function is used, performing a reset:
0: Cancels the synchronization mode (G81 or G81.5).
1: Does not cancel the synchronization mode. The mode is canceled only by the G80 or G80.5 command.

#2 HDR Direction of helical gear compensation (usually, set 1.)
(Example) To cut a left-twisted helical gear when the direction of rotation about the C-axis is the negative (-) direction:
0: Set a negative (-) value in P.
1: Set a positive (+) value in P.

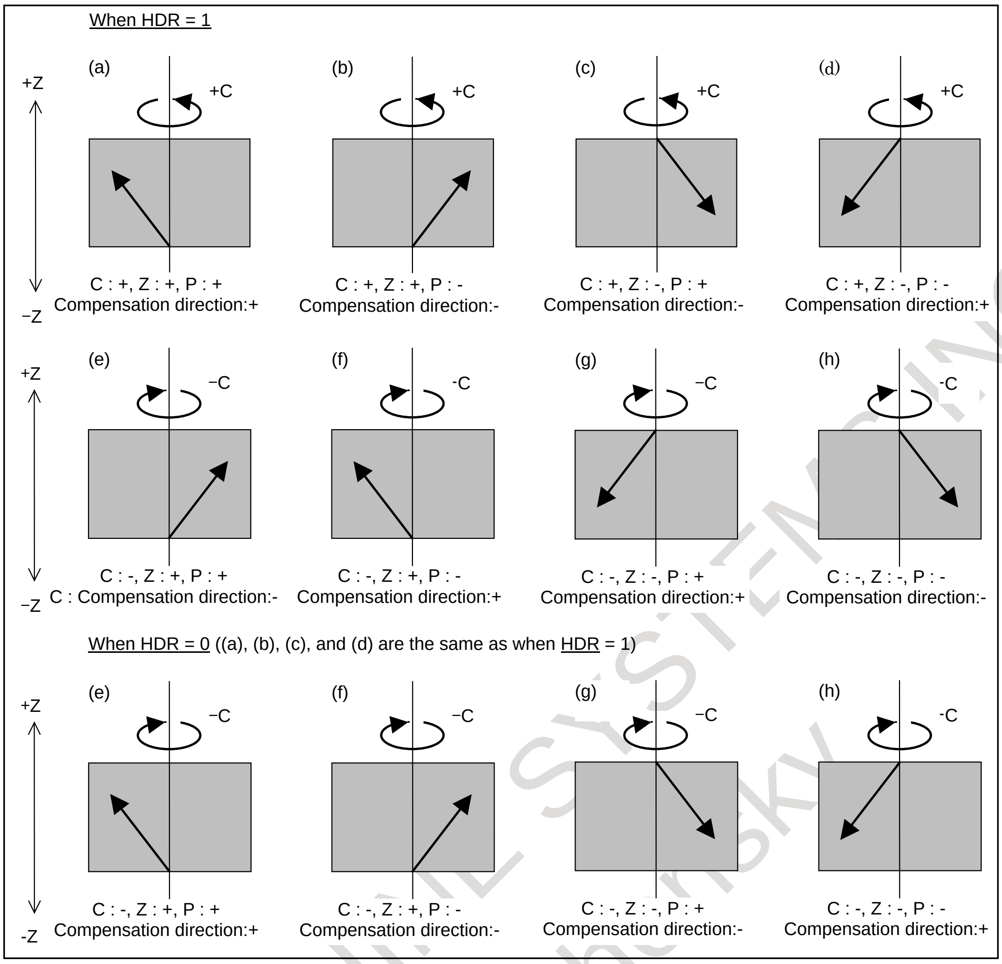


Fig. 1.9.1 (e) Direction of helical gear compensation

	#7	#6	#5	#4	#3	#2	#1	#0
7701					**LZR**			

[Input type] Parameter input
[Data type] Bit path

#3 LZR When L (number of hob threads) = 0 is specified at the start of EGB synchronization (G81):
0: Synchronization is started, assuming that L = 1 is specified.
1: Synchronization is not started, assuming that L = 0 is specified. However, helical gear compensation is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
7702					**ART**			**TDP**

[Input type] Parameter input
[Data type] Bit path

#0 TDP The specifiable number of teeth, T, of the electronic gear box (G81) is:
0: 1 to 5000
1: 0.1 to 500 (1/10 of a specified value)

> **NOTE**
> In either case, a value from 1 to 5000 can be specified.

#3 ART The retract function executed when an alarm is issued is:
0: Disabled.
1: Enabled.
When an alarm is issued, a retract operation is performed with a set feedrate and travel distance (parameters Nos. 7740 and 7741).

NOTE
If a servo alarm is issued for other than the axis along which a retract operation is performed, the servo activating current is maintained until the retract operation is completed.

	#7	#6	#5	#4	#3	#2	#1	#0
7703						ARO	ARE	ERV

[Input type] Parameter input
[Data type] Bit path

#0 ERV During EGB synchronization (G81), feed per revolution is performed for:
0: Feedback pulses.
1: Pulses converted to the speed for the workpiece axis.

#1 ARE The retract function executed when an alarm is issued retracts the tool during:
0: EGB synchronization or automatic operation (automatic operation signal OP <Fn000.7> = “1”).
1: EGB synchronization.

#2 ARO The retract function executed when an alarm is issued retracts the tool during:
0: EGB synchronization.
1: EGB synchronization and automatic operation (automatic operation signal OP <Fn000.7> = “1”).

NOTE
This parameter is valid when bit 1 (ARE) of parameter No. 7703 is set to 1.

The table lists the parameter settings and corresponding operation.

ARE	ARO	Operation
1	0	During EGB synchronization
1	1	During EGB synchronization and automatic operation
0	0	During EGB synchronization or automatic operation
0	1	

NOTE
Parameters ARE and ARO are valid when bit 3 (ART) of parameter No. 7702 is set to 1 (when the retract function executed when an alarm is issued).

7709	Number of the axial feed axis for helical gear compensation

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to Number of controlled axes
This parameter sets the number of the axial feed axis for a helical gear.

> **NOTE**
> When this parameter is set to 0 or a value outside the valid setting range, the Z-axis becomes the axial feed axis.

	#7	#6	#5	#4	#3	#2	#1	#0
7731	HAD		HBR		ECN			EFX

[Input type] Parameter input
[Data type] Bit path

#0 EFX As the EGB command:
0: G80 and G81 are used.
1: G80.4 and G81.4 are used.

> **NOTE**
> When this parameter is set to 0, no canned cycle for drilling can be used.

#3 ECN When the automatic phase synchronization function for the electronic gear box is disabled, during EGB synchronization, the G81 or G81.5 command:
0: Cannot be issued again. (The alarm PS1595, "ILL-COMMAND IN EGB MODE" is issued.)
1: Can be issued again.

#5 HBR In EGB synchronization start command G81.4, the number of teeth is:
0: Specified in T.
1: Specified in R.

> **NOTE**
> This parameter is valid when bit 0 (EFX) of parameter No. 7731 is set to 1.

#7 HAD In electronic gear box, the timing for reflecting helical gear compensation and travel distance of automatic phase synchronization to absolute coordinates is:
0: When synchronization is canceled.
1: During helical gear compensation and automatic phase synchronization.

7740	Feedrate during retraction

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the feedrate during retraction for each axis.

7741	Retract amount

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the retract amount for each axis.

7772	Number of position detector pulses per rotation about the tool axis

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[Valid data range] 1 to 999999999

This parameter sets the number of pulses per rotation about the tool axis (master axis), for the position detector.
For an A/B phase detector, set this parameter with four pulses equaling one A/B phase cycle.

7773	Number of position detector pulses per rotation about the workpiece axis

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[Valid data range] 1 to 999999999

This parameter sets the number of pulses per rotation about the workpiece axis (slave axis), for the position detector.
Set the number of pulses output by the detection unit.
Set parameters Nos. 7772 and 7773 when using the G81 EGB synchronization command.

[Example 1] When the EGB master axis is the spindle and the EGB slave axis is the C-axis

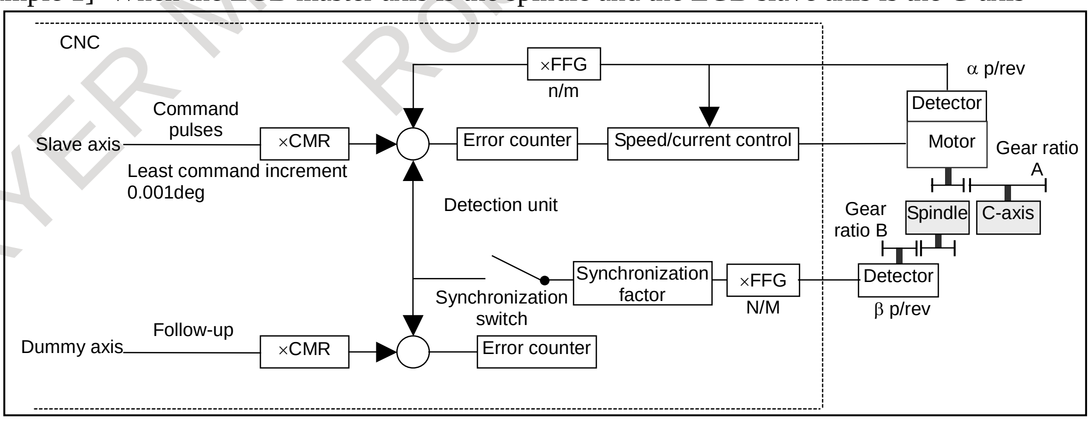


Fig. 1.9.1 (f)

Gear ratio of the spindle to the detector B:
1/1 (The spindle and detector are directly connected to each other.)
Number of detector pulses per spindle rotation β: 80,000 pulses/rev
(Calculated for four pulses for one A/B phase cycle)
FFG N/M of the EGB dummy axis : 1/1

Gear ratio of the C-axis A :1/36 (One rotation about the C-axis to 36 motor rotations)
Number of detector pulses per C-axis rotation α: 1,000,000 pulses/rev
C-axis CMR: 1
C-axis FFG n/m :1/100

In this case, the number of pulses per spindle rotation is:
$80000 \times 1/1 = 80000$
Therefore, set 80000 for parameter No. 7772.
The number of pulses per C-axis rotation in the detection unit is:

$$1000000 \div \frac{1}{36} \times \frac{1}{100} = 360000$$

Therefore, set 360000 for parameter No. 7773.

[Example 2] When the gear ratio of the spindle to the detector B is 2/3 for the above example (When the detector rotates twice for three spindle rotations)
In this case, the number of pulses per spindle rotation is:

$$80000 \times \frac{2}{3} = \frac{160000}{3}$$

160000 cannot be divided by 3 without a remainder. In this case, change the setting of parameter No. 7773 so that the ratio of the settings of parameters Nos. 7772 and 7773 indicates the value you want to set.

$$\frac{\text{No.7772}}{\text{No.7773}} = \frac{160000/3}{360000} = \frac{160000}{360000 \times 3} = \frac{160000}{1080000}$$

Therefore, set 160000 for parameter No. 7772 and 1080000 for parameter No. 7773.
As described above, all the settings of parameters Nos. 7772 and 7773 have to do is to indicate the ratio correctly. So, you can reduce the fraction indicated by the settings. For example, you may set 16 for parameter No. 7772 and 108 for parameter No. 7773 for this case.

	#7	#6	#5	#4	#3	#2	#1	#0
2273		EGFx						

[Input type] Parameter input
[Data type] Bit axis

#6 EGFx FFG is:
0: Not considered in the number of pulses of the position detector per rotation about the master axis.
1: Considered.
Set for the EGB dummy axis.

> **NOTE**
> EGFx is valid for A/B phase type position detector. In the case of using serial type position detector, FFG is always considered.

The synchronization coefficient is subject to the following restriction:

$$\text{Synchronization coefficient} = \frac{K_n}{K_d} = \frac{L}{T} \times \frac{\beta}{\alpha}$$

$-2147483648 \le K_n \le 2147483647$
$1 \le K_d \le 2147483647$

If above condition cannot be satisfied, set this parameter bit to 1. With this setting, FFG is considered in the number of pulses of the position detector per rotation about the master axis, and by selecting FFG appropriately, it is possible to be satisfied above condition without the change of synchronization coefficient. $\alpha[\text{New}] = \alpha[Old] \times \frac{N}{M}$

N: Numerator of FFG
M: Denominator of FFG

[Setting example]

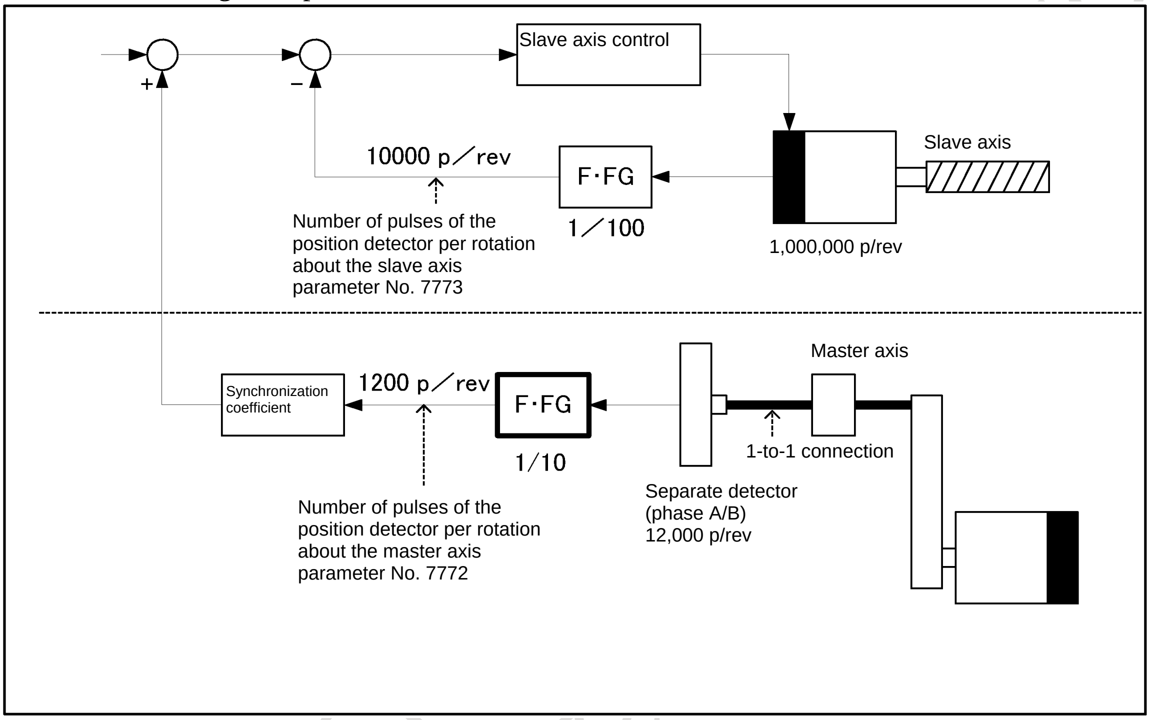


Fig. 1.9.1 (g)

Master axis conditions:
The separate detector must be 12000 p/rev.
The master axis and the separate detector must have a 1-to-1 connection.

In this example, EGFx (bit6 of parameter No.2273) is set to 1, FFG is set to 1/10, and the number of pulses of the position detector per rotation about the master axis is
$\alpha = 12000 \times \frac{1}{10} = 1200$

	#7	#6	#5	#4	#3	#2	#1	#0
4399						**SOSALWs**		

[Input type] Parameter input
[Data type] Bit spindle

#2 SOSALWs Soft start/stop function is:
0: Disabled if *ESP is ”0” (emergency stop) or if MRDY is ”0”.
1: Enabled also if *ESP is ”0” (emergency stop) or if MRDY is ”0”.

4030	**Soft start/stop setting time**

[Input type] Parameter input
[Data type] Word spindle

[Unit of data] 1min^{-1}/sec
[Valid data range] 0 to 32767

This parameter sets an acceleration (rate at which the speed changes) applied when the soft start/stop function is enabled (soft start/stop signal SOCNA <Gn071> is “1”).

4508	Rate of change in acceleration at soft start/stop

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 10min^{-1}/sec^{2}
[Valid data range] 0 to 32767

This parameter sets the rate of change of acceleration (rate at which the acceleration changes) applied when the soft start/stop function is enabled (soft start/stop signal SOCNA <Gn071> is “1”).

> **NOTE**
> If the setting is 0, speed commands are linear when the soft start/stop function is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
4472						SOSALWs		

[Input type] Parameter input
[Data type] Bit spindle

#2 SOSALWs Soft start/stop function(if the spindle axis is a sub axis of spindle switching control) is:
0: Disabled if *ESP is ”0” (emergency stop) or if MRDY is ”0”.
1: Enabled also if *ESP is ”0” (emergency stop) or if MRDY is ”0”.

Alarm and message

Number	Message	Description
PS1593	EGB PARAMETER SETTING ERROR	Error in setting a parameter related to the EGB (1) The setting of bit 0 (SYNx) of parameter No. 2011, is not correct. (2) The slave axis specified with G81 is not set as a rotation axis. (bit 0 (ROTx) of parameter No. 1006) (3) Number of pulses per rotation (parameter No. 7772 or 7773) is not set.
PS1594	EGB FORMAT ERROR	Error in the format of the block of an EGB command (1) T (number of teeth) is not specified in the G81 block. (2) In the G81 block, the data specified for one of T, L, P, and Q is out of its valid range. (3) In the G81 block, only one of P and Q is specified.
PS1595	ILL-COMMAND IN EGB MODE	During synchronization with the EGB, a command that must not be issued is issued. (1) Slave axis command using G27, G28, G29, G30, G33, G53, etc. (2) Inch/metric conversion command using G20, G21, etc. (3) Synchronization start specified by G81 when bit 3 (ECN) of parameter No. 7731 is set to 0
PS1596	EGB OVERFLOW	An overflow occurred in the calculation of the synchronization coefficient.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Electronic gear box

1.9.2 Spindle Electronic Gear Box

Overview

In a system in which two spindles are used as the tool axis and workpiece axis, a gear can be machined (ground or cut) by synchronizing the workpiece axis rotation with the tool axis (grinding wheel or hob) rotation.

To synchronize these two spindles, the spindle electronic gear box is used (the electronic gear box is hereinafter called the EGB). In the spindle EGB, synchronization pulses are generated based on feedback pulses from the position detector attached to the tool axis (master axis), and the workpiece axis (slave axis) rotates based on the synchronization pulses. Feedback pulses are transferred from the master axis to the slave axis by communication between amplifiers.

Specification

The specification of spindle EGB synchronization are as follows:

(1) Spindle EGB synchronization is started by specifying the T command (number of teeth) and L command (number of hob threads), which determine the synchronization coefficient, in the G81 block. The synchronization is canceled by specifying G80.
(2) The synchronization coefficient is calculated using the T command (number of teeth) and L command (number of hob threads) in the G81 block, and the number of position detector pulses per rotation about each of the tool and workpiece pulses (set in the relevant parameter).
(3) This function allows a retract operation.
(4) A helical gear can be cut by specifying the Q command (module or diametral pitch) and P command (gear helix angle) in the G81 block.
(5) During EGB synchronization, the synchronization relationship is maintained regardless of whether the operation is automatic or manual.
(6) Spindle amplifier SPM type B is required for both the master and slave axes. In addition, the spindle amplifiers can have a 1-to-1 connection only. For details, refer to "SPINDLE EGB (SPINDLE ELECTRONIC GEAR BOX)" in "FANUC AC SPINDLE MOTOR α*i* series, FANUC AC SPINDLE MOTOR β*i* series, FANUC BUILT-IN SPINDLE MOTOR β*i* series PARAMETER MANUAL (B-65280EN)".
(7) To turn the EGB synchronization mode on, the slave axis must be put in the Cs contour control mode, though the master axis may be in any control mode.

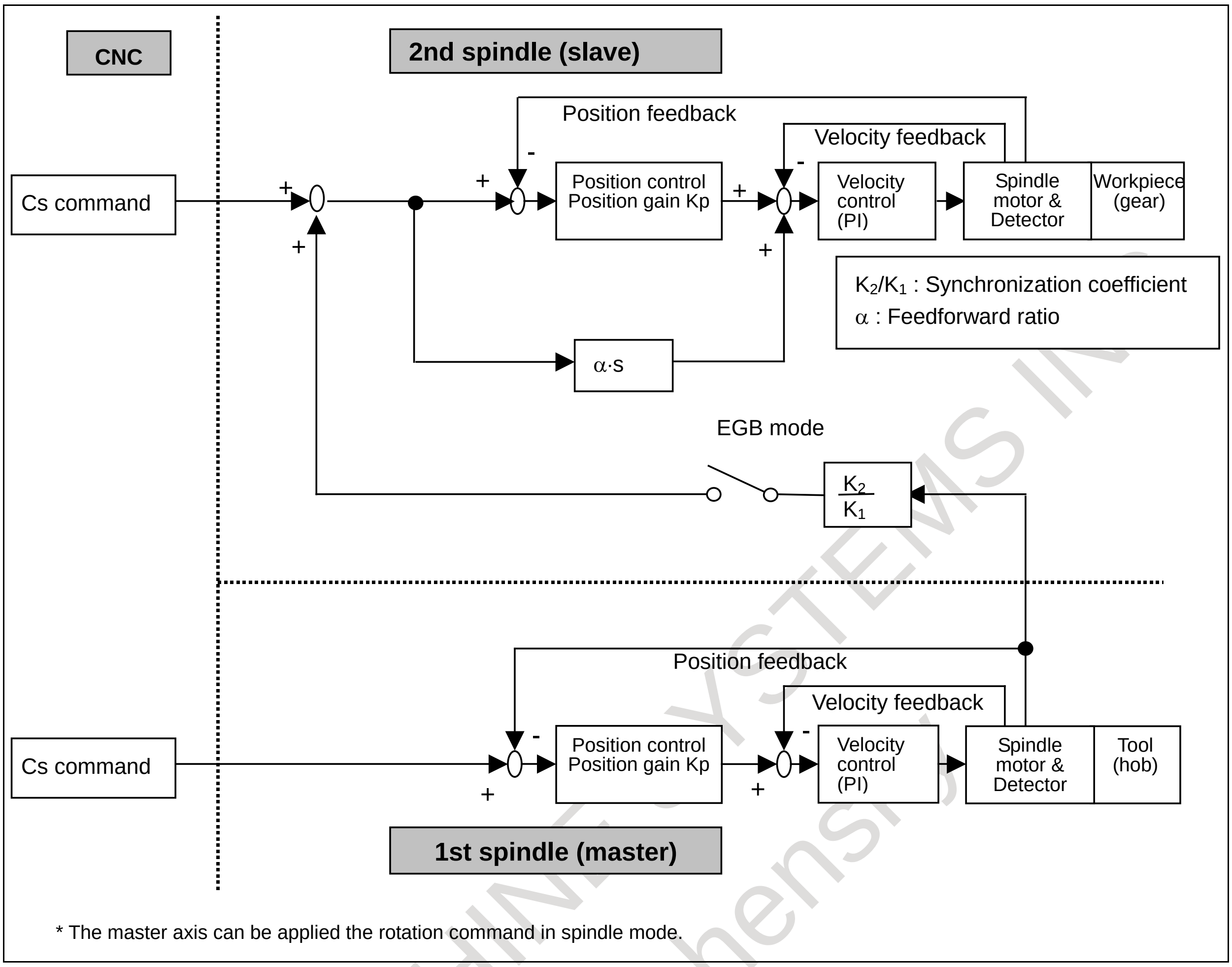


Fig. 1.9.2 (a) Block diagram of spindle EGB control

Format

	Bit 0 (EFX) of parameter No.7731=0	Bit 0 (EFX) of parameter No.7731=1	
		Bit 5 (HBR) of parameter No.7731=1	Bit 5 (HBR) of parameter No.7731=0
Start of synchronization	**G81 T_ (L_) (Q_ P_) ;**	**G81.4 R_ (L_) (Q_ P_) ;**	**G81.4 T_ (L_) (Q_ P_) ;**
Cancellation of synchronization	**G80 ;**	**G80.4 ;**	**G80.4 ;**
	(*1) (*4)	**(*2) (*4)**	**(*3) (*4)**

T(or R) : Number of teeth (Specifiable range: 1 to 5000)
L : Number of hob threads (Specifiable range: -250 to 250)
The sign of L determines the direction of rotation for the workpiece axis.
When L is positive, the direction of rotation for the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation for the workpiece axis is negative (- direction).
When L is 0, it follows the setting of bit 3 (LZR) of parameter No.7701.
If L is not specified, the number of hob threads is assumed 1.
Q : Module or diametral pitch
Specify a module in the case of metric input.
(Unit: 0.00001mm, Specifiable range: 0.01 to 25.0mm)
Specify a diametral pitch in the case of inch input.
(Unit: $0.00001inch^{-1}$, Specifiable range: 0.01 to $254.0\ inch^{-1}$)
P : Gear helix angle
(Unit: 0.0001deg, Specifiable range: -90.0 to 90.0deg)
*1 Use it for machining centers.
*2 Use it for lathes.
*3 Use it for machining centers.
This format enables specification of the same G codes as those for lathes.
*4 When specifying Q and P, the user can use a decimal point.

NOTE
Specify G81, G80, G81.4, and G80.4 in a single block.

Explanation

- Parameter setting

For spindle EGB control, the following parameters must be set:

- Controlled axis number for the slave axis (parameter No. 7710)
- Number of position detector pulses per rotation about the tool axis (parameter No. 7772)
- Number of position detector pulses per rotation about the workpiece axis (parameter No. 7773)
- Spindle EGB master axis enable (bit 7 of parameter No. 4352)
- Spindle EGB slave axis enable (bit 6 of parameter No. 4352)
- Number of sinusoidal waves from the master spindle position detector (parameter No. 4386)

- Start/Cancellation of synchronization

When rotation about the tool axis (master axis) starts after G81 is specified, EGB synchronization starts according to the synchronization relationship specified in the G81 block, and rotation about the workpiece axis (slave axis) starts. When EGB synchronization starts, the EGB mode signal SYNMOD <Fn065.6> becomes “1”.

When rotation about the tool axis is stopped, rotation about the workpiece axis is also stopped. At this time, specifying G80 cancels EGB synchronization. When EGB synchronization is canceled, the EGB mode signal SYNMOD <Fn065.6> becomes 0.

Specify P and Q to use helical gear compensation.

If only either P or Q is specified, alarm PS1594 “EGB FORMAT ERROR” is issued.

G81 cannot be specified again during EGB synchronization. In addition, the specification of T, L, Q, and P cannot be modified during synchronization. Start and cancel synchronization when rotation about the tool axis (master axis) stops.

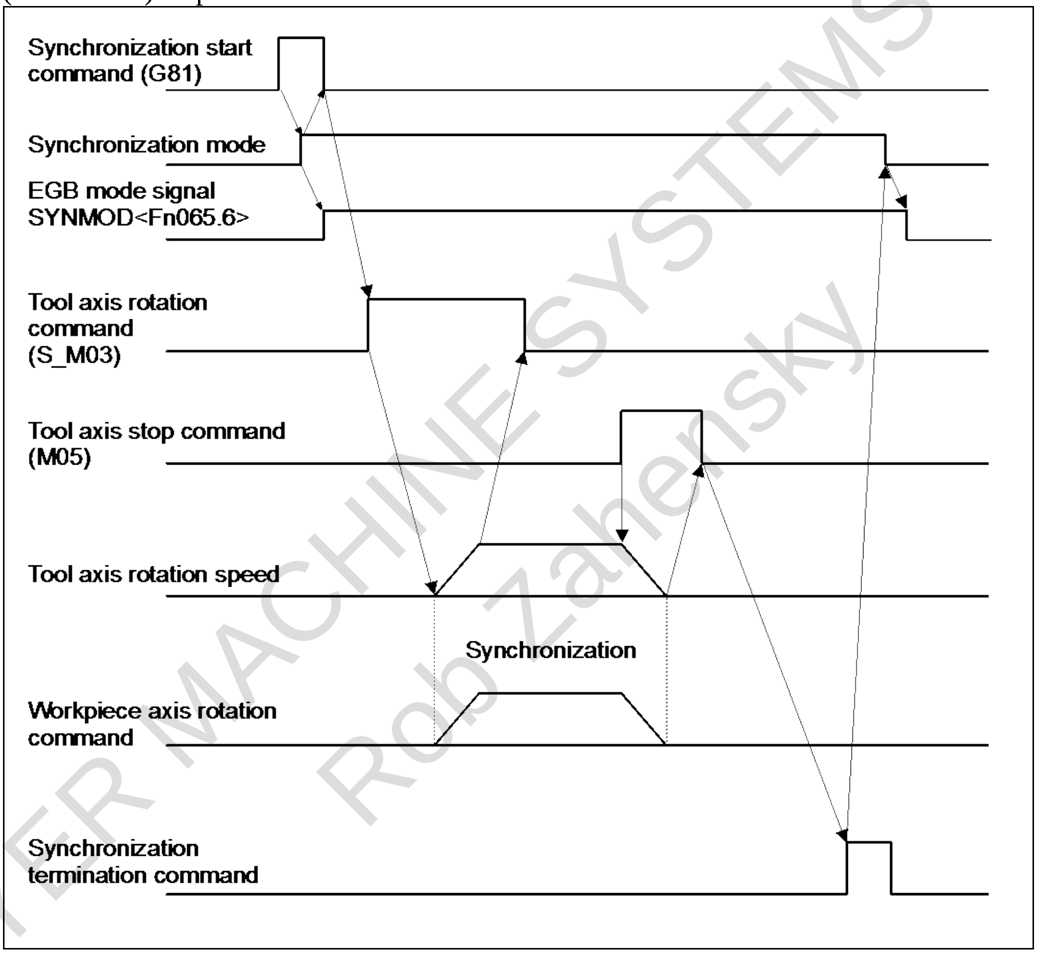


Fig. 1.9.2 (b) Synchronization start/cancellation timing chart

> ⚠ **CAUTION**
> 1 Feed hold, interlock, and machine lock are invalid to a slave axis in EGB synchronization.
> 2 Even if an OT alarm is issued for a slave axis in EGB synchronization, synchronization will not be canceled.
> 3 During synchronization, it is possible to execute a move command for a slave axis and other axes, using a program. The move command for a slave command must be an incremental one.

NOTE

1 If bit 0 (HBR) of parameter No. 7700 is set to 1, EGB synchronization will not be canceled due to a reset. Usually, set this parameter bit to 1.

2 Start and cancel EGB synchronization when rotation about the master and slave axes stops. It means that rotation about the master axis should be started while the EGB mode signal SYNMOD is “1” (see Fig. 1.10.2 (b), "Synchronization start/cancellation timing chart"). If the master axis starts rotating before the EGB mode signal SYNMOD becomes “1”, synchronization is not performed correctly.

3 Perform the reference position return of the Cs contour controlled axis for the master and slave axes before specifying G81. During synchronization, reference position return cannot be performed. Do not place the machine in the reference position return mode during synchronization.

4 If a parameter for axis setting (No. 7710 or 4352) is not set correctly, alarm PS1593, “EGB PARAMETER SETTING ERROR” is issued when G81 is specified.

5 In synchronization mode, it is not possible to specify G27, G28, G29, G30, and G53 for a slave axis.

6 In EGB synchronization mode, AI contour control mode is temporarily canceled.

7 The position display of the slave axis is updated based on synchronization pulses as follows:
   - During synchronization, only the machine coordinate is updated. The absolute and relative coordinates are not updated.
   - When synchronization is canceled, the amount of travel during synchronization, as rounded to 360-degree units is added to the absolute coordinate.

8 The direction of rotation about the slave axis depends on the direction of rotation about the master axis. When the direction of rotation about the master axis is positive, the direction of rotation about the slave axis is also positive; when the direction of rotation about the master axis is negative, the direction of rotation about the slave axis is also negative. A negative value can be specified for L to make the direction of rotation about the slave axis opposite to the direction of rotation about the master axis, however.

9 The synchronization mode is canceled by a servo alarm, spindle alarm, PW alarm, or emergency stop.

10 Synchronization is not maintained if the slave axis is in the servo off state.

11 During synchronization, manual handle interrupt can be performed on the workpiece and other axes.

12 In synchronization mode, no inch/metric conversion commands (G20 and G21) cannot be issued.

13 If bit 0 (EFX) of parameter No. 7731 is 0, no canned cycle for drilling can be used. To use a canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.

14 If bit 0 (TDP) of parameter No. 7702 is 1, the permissible range of T is 0.1 to 500 (1/10 of the specified value).

15 If, at the start of EGB synchronization (G81), L is specified as 0, synchronization starts with L assumed to be 1 if bit 3 (LZR) of parameter No.7701 is 0; if bit 3 (LZR) of parameter No.7701 is 1, synchronization is not started with L assumed to be 0. At this time, helical gear compensation is performed.

NOTE

16 Feed per revolution is performed on the feedback pulses on the spindle. By setting bit 0 (ERV) of parameter No. 7703 to 1, feed per revolution can be performed based on the speed on the synchronous slave axis.
17 Actual cutting feedrate display does not take synchronization pulses into consideration.
18 For an EGB slave axis, synchronous and composite control cannot be executed.
19 The G81 command cannot be specified to use the servo EGB and spindle EGB together. To use them together, specify the G81 command for the spindle EGB and the G81.5 command for the servo EGB.
20 When the simple spindle EGB function is used, this function cannot be used. If the G81 command is specified for the slave axis of the simple spindle EGB, alarm PS1593 “EGB PARAMETER SETTING ERROR” is issued.
21 The master axis cannot be shared by the simple spindle EGB function and this function.
22 Not advanced preview feed-forward but conventional feed-forward is enabled in the path where EGB synchronization mode is effective.

- Program example

Axis configuration X, Y, Z, B (Cs axis: tool axis/master axis), C (Cs axis: workpiece axis/slave axis)

```
O1000 ;
N0010 G80 ;
N0020 G28 G91 B0 C0 ;   Performs reference position return for the tool and workpiece axes.
N0030 G81 T20 L1 ;      Starts synchronization.
N0040 Mxx ;             Rotates the tool axis by the constant speed command of PMC axis control.
N0050 G04 X1000 ;       Waits until rotation of the tool axis becomes constant.
N0060 G01 X_ F_ ;       Moves the tool along the X-axis (cutting).
N0070 G01 Z_ F_ ;       Moves the tool along the Z-axis (machining).
-------------------------
-------------------------
N0100 G01 X_ F_ ;       Moves the tool along the X-axis (retraction).
N0110 M05 ;             Stops rotation about the tool axis.
N0120 G80 ;             Cancels synchronization.
N0130 M30 ;
```

- Helical gear compensation

For a helical gear, the workpiece axis is compensated for the movement along the Z-axis (axial feed axis) based on the torsion angle of the gear.
Helical gear compensation is performed with the following formulas:
For a helical gear, the workpiece axis is compensated for the movement along the Z-axis (axial feed axis) based on the torsion angle of the gear.
Helical gear compensation is performed with the following formulas:

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (for metric input)}$$

$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (for inch input)}$$

where

Compensation angle: Signed absolute value (deg)

- Z : Amount of travel on the Z-axis after the specification of G81
- P : Signed gear helix angle (deg)
- π : Circular constant
- T : Number of teeth
- Q : Module (mm) or diametral pitch ($inch^{-1}$)

Use P, T, and Q specified in the G81 block.
In helical gear compensation, the machine coordinates on the workpiece axis and the absolute coordinates are updated with helical gear compensation.
The updated timing is as follows.

<table>
<tr><th>Machine coordinates</th><th colspan="2">Absolute coordinates</th></tr>
<tr><td rowspan="3">During helical gear compensation</td><td colspan="2">Bit 7(HAD) of parameter No.7731</td></tr>
<tr><td>= 0</td><td>= 1</td></tr>
<tr><td>Synchronization cancel</td><td>During helical gear compensation</td></tr>
</table>

- Direction of helical gear compensation

The direction depends on bit 2 (HDR) of parameter No. 7700.

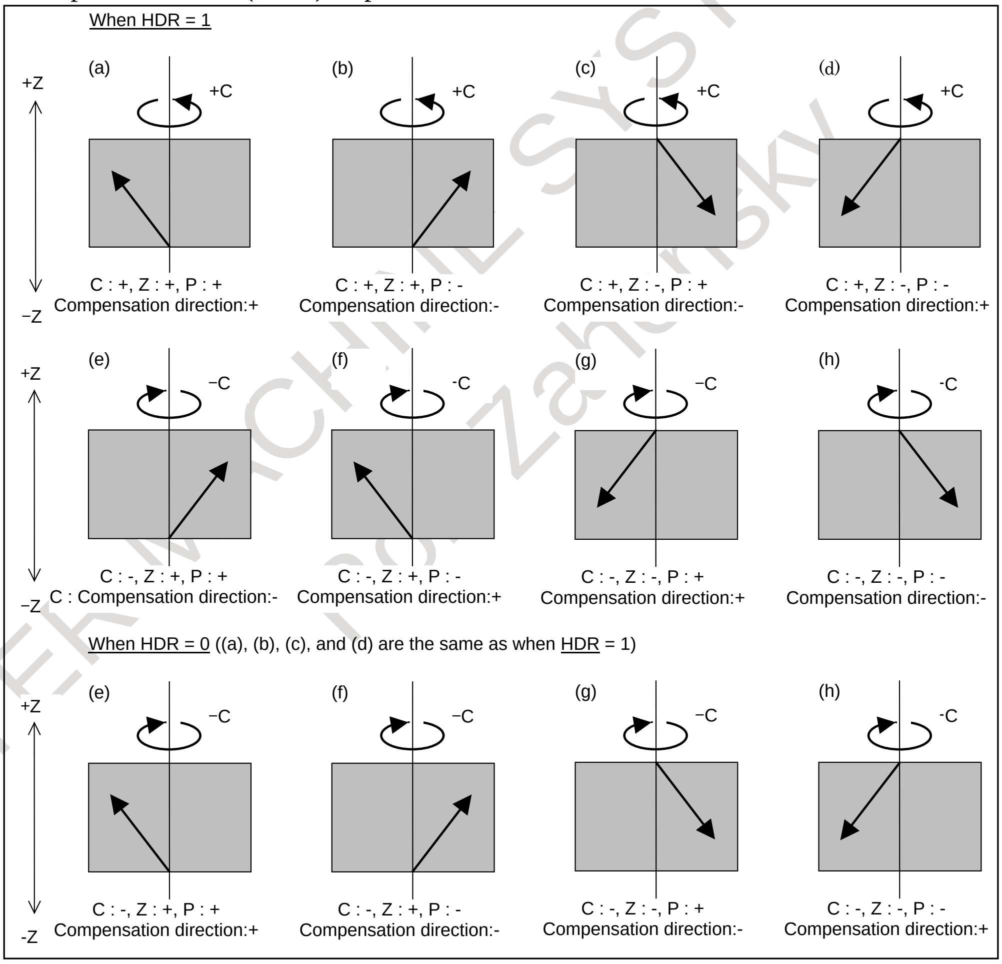


Fig. 1.9.2 (c) Direction of helical gear compensation

- Synchronization coefficient

A synchronization coefficient is internally represented using a fraction (K_2/K_1) to eliminate an error. The formula below is used for calculation.

$$\text{Synchronization coefficient} = \frac{K_2}{K_1} = \frac{L}{T} \times \frac{\beta}{\alpha}$$

where

L : Number of hob threads
T : Number of teeth
α : Number of pulses of the position detector per rotation about the master axis (parameter No. 7772)
β : Number of pulses of the position detector per rotation about the slave axis (parameter No. 7773)

K_2/K_1 is a value resulting from reducing the right side of the above formula, but the result of reduction is subject to the following restrictions:

$-32767 \le K_2 \le 32767$

$1 \le K_1 \le 32767$

When this restriction is not satisfied, the alarm PS1596 “EGB OVERFLOW” is issued when G81 is specified.

The values of K_2 and K_1 are set in parameters Nos. 4387 and 4388 automatically when G81 is specified. If T is not specified in the G81 block, alarm PS1594 “EGB FORMAT ERROR” is issued. If L is not specified in that block, the synchronization coefficient is calculated, assuming L = 1.

Example)

When the number of pulses that correspond to one rotation (360000) is specified for the tool axis (master axis) under the following conditions, the position command pulses are distributed as shown in Fig. 1.10.2 (d).

Number of hob threads L: 10
Number of teeth T: 100
Number of pulses of the position detector per rotation about the tool axis : 360000
Number of pulses of the position detector per rotation about the workpiece axis : 360000

$$\text{Synchronization coefficient} = \frac{K_2}{K_1} = \frac{L}{T} \times \frac{\beta}{\alpha} = \frac{10}{100} \times \frac{360000}{360000} = \frac{1}{10}$$

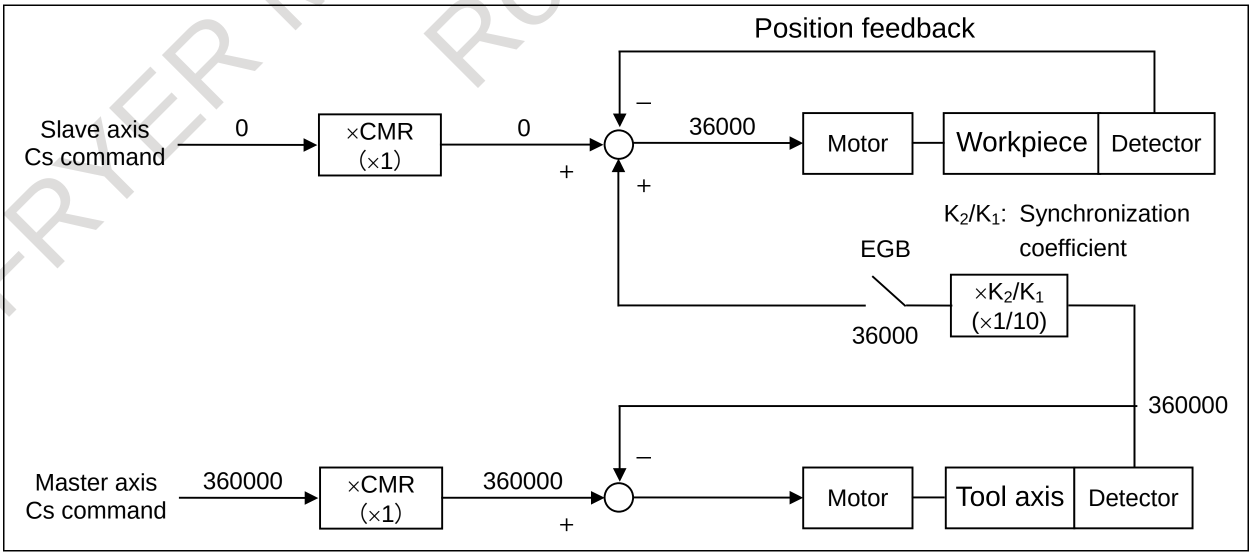


Fig. 1.9.2 (d) Command pulse distribution

As shown in Fig. 1.10.2 (d), when 360000 pulses (number of pulses required for one rotation about the master axis) are specified for the master axis, the value of the position command to the slave axis by EGB is obtained by multiplying the number of pulses required for one rotation about the slave axis by the ratio of the number of hob threads to the number of teeth (rotation ratio of the slave axis to the master axis):

360000 × 1/10 = 36000

・Retract function

See Item, “Retract function” in the Subsection “Electronic Gear Box”.

Signal

EGB mode signal SYNMOD <Fn065.6>

[Classification] Output signal

[Function] Posts notification that synchronization using the EGB is in progress.

[Operation] This signal is set to “1” in the following case:

- While synchronization using the EGB is in progress

This signal is set to “0” in the following case:

- Once synchronization using the EGB has terminated

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn065		SYNMOD						

Parameter

The table below gives parameters related to spindle EGB.

Parameter number	Description
1006#0 1006#1	An EGB slave axis requires that the setting of a rotary axis (type A) (bit 0 (ROT) of parameter No. 1006 be 1 and bit 1 (ROS) of parameter No. 1006 be 0.
4036	Feed forward coefficient for serial spindle
4352#4	Feed forward setting
4352#6	Setting of the spindle EGB slave axis
4352#7	Setting of the spindle EGB master axis
4386	Number of sinusoidal waves from the master spindle position detector
4387	Numerator of synchronization coefficient
4388	Denominator of synchronization coefficient
7700#0	The synchronous mode is canceled (0)/not canceled (1) by a reset.
7700#2	Compensation direction for helical gear compensation
7701#3	At the start of synchronization (G81), synchronization is started (0)/not started (1) if the number of hob threads L is specified as 0.
7702#0	The specifiable number of teeth, T, at the start of synchronization (G81) is not reduced to a 1/10 of a specified value (0)/reduced (1).
7702#3	The retract function with an alarm is disabled (0)/enabled (1).
7703#0	During synchronization (G81), feed per revolution is performed for feedback pulses (0)/pulses converted to the speed for the workpiece axis(1).
7703#1,#2	Specify when to perform a retract operation with the retract function with an alarm; during synchronization; during synchronization and automatic operation; or during synchronization or automatic operation.
7709	Number of the axial feed axis in helical gear compensation
7710	The controlled axis number for the spindle EGB slave axis
7731#0	The EGB command is G80 and G81(0)/G80.4 and G81.4(1).
7731#5	In EGB synchronization start command G81.4, the number of teeth is specified in T (0)/specified in R (1).
7740	Retraction speed

Parameter number	Description
7741	Retract amount
7772	Number of position detector pulses per rotation about tool axis
7773	Number of position detector pulses per rotation about workpiece axis

4036	Feed forward coefficient for serial spindle

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%
[Valid data range] 0 to 10000

This parameter sets the feed forward coefficient in the Cs axis contour control mode.
When the setting is smaller than or equal to 100: In units of 1%
When the setting is greater than 100: In units of 0.01%
Set 100 for the slave spindle.

	#7	#6	#5	#4	#3	#2	#1	#0
4352	SPEGBM	SPEGBS		FFALWS				

[Input type] Parameter input
[Data type] Bit spindle

#4 FFALWS Feed forward setting
0: Feed forward is enabled only in cutting feed.
1: Feed forward is always enabled.
Set 1 for the slave spindle.

#6 SPEGBS The spindle EGB function for slave spindle is:
0: Disabled.
1: Enabled.
Set 1 for the slave spindle.

#7 SPEGBM The spindle EGB function for master spindle is:
0: Disabled.
1: Enabled.
Set 1 for the master spindle.

4386	Number of sinusoidal waves from the master spindle position detector

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1λ/rev
[Valid data range] 0, 64 to 4096

This parameter sets the number of sinusoidal waves per spindle rotation from the master spindle position detector.
Set this parameter for the slave spindle amplifier.
Setting 0 in this parameter is equivalent to setting the synchronization coefficient to 0.

4387	Numerator of synchronization coefficient

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] -32767 to 32767

The numerator of the synchronization coefficient is set in this parameter automatically when G81 is specified.

4388	Denominator of synchronization coefficient

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 1 to 32767

The denominator of the synchronization coefficient is set in this parameter automatically when G81 is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
7700						HDR		HBR

[Input type] Parameter input
[Data type] Bit path

#0 HBR When the electronic gear box (EGB) function is used, performing a reset:
0: Cancels the synchronous mode (G81 or G81.5).
1: Does not cancel the synchronous mode. The mode is canceled only by the G80 or G80.5 command.

#2 HDR Direction of helical gear compensation (usually, set 1.)
(Example) To cut a left-twisted helical gear when the direction of rotation about the C-axis is the negative (-) direction:
0: Set a negative (-) value in P.
1: Set a positive (+) value in P.

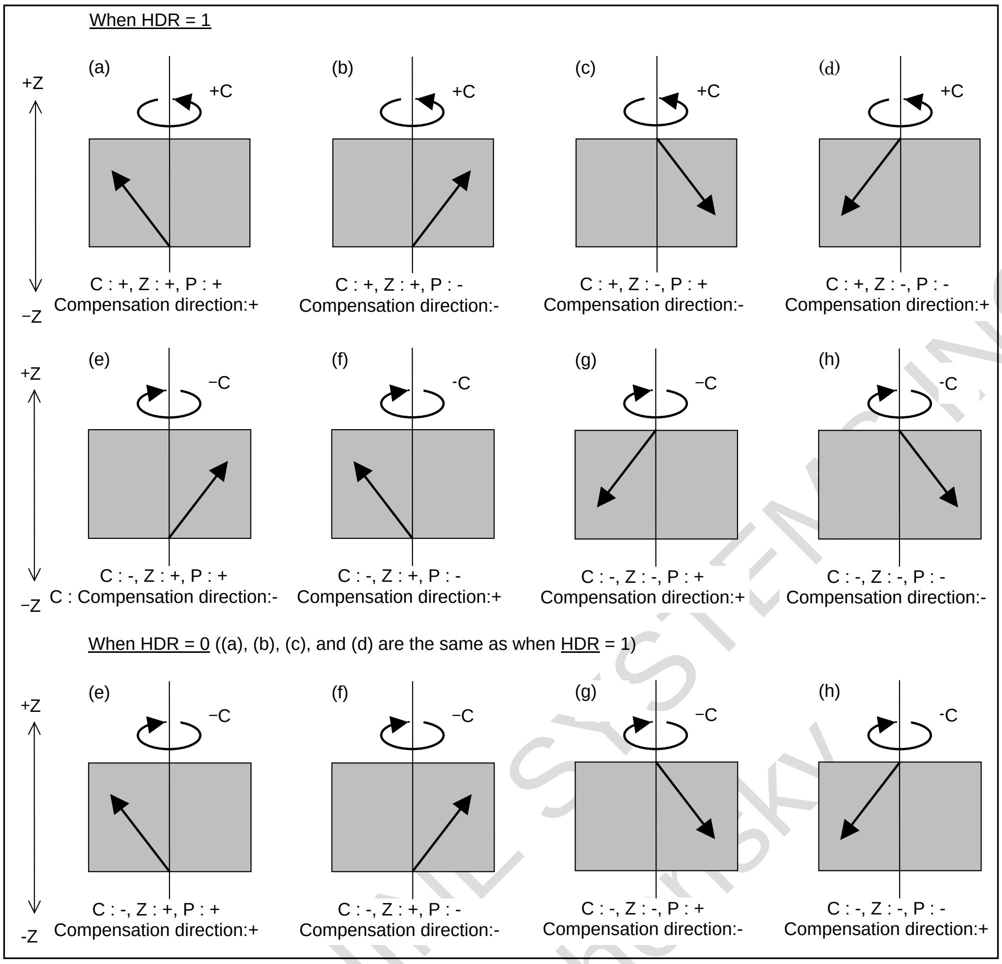


Fig. 1.9.2 (e) Direction of helical gear compensation

	#7	#6	#5	#4	#3	#2	#1	#0
7701					LZR			

[Input type] Parameter input
[Data type] Bit path

#3 LZR When L (number of hob threads) = 0 is specified at the start of EGB synchronization (G81):
0: Synchronization is started, assuming that L = 1 is specified.
1: Synchronization is not started, assuming that L = 0 is specified. However, helical gear compensation is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
7702					ART			TDP

[Input type] Parameter input
[Data type] Bit path

#0 TDP The specifiable number of teeth, T, of the electronic gear box (G81) is:
0: 1 to 5000
1: 0.1 to 500 (1/10 of a specified value)

> **NOTE**
> In either case, a value from 1 to 5000 can be specified.

#3 ART The retract function executed when an alarm is issued is:
0: Disabled.
1: Enabled.
When an alarm is issued, a retract operation is performed with a set feedrate and travel distance (parameters Nos. 7740 and 7741).

> **NOTE**
> If a servo alarm is issued for other than the axis along which a retract operation is performed, the servo activating current is maintained until the retract operation is completed.

	#7	#6	#5	#4	#3	#2	#1	#0
7703						ARO	ARE	ERV

[Input type] Parameter input
[Data type] Bit path

#0 ERV During EGB synchronization (G81), feed per revolution is performed for:
0: Feedback pulses.
1: Pulses converted to the speed for the workpiece axis.

#1 ARE The retract function executed when an alarm is issued retracts the tool during:
0: EGB synchronization or automatic operation (automatic operation signal OP <Fn000.7> = “1”).
1: EGB synchronization.

#2 ARO The retract function executed when an alarm is issued retracts the tool during:
0: EGB synchronization.
1: EGB synchronization and automatic operation (automatic operation signal OP = “1”).

> **NOTE**
> This parameter is valid when bit 1 (ARE) of parameter No. 7703 is set to “1”.

The following table lists the parameter settings and corresponding operation.

<table>
<tr><th>ARE</th><th>ARO</th><th>Operation</th></tr>
<tr><td>1</td><td>0</td><td>During EGB synchronization</td></tr>
<tr><td>1</td><td>1</td><td>During EGB synchronization and automatic operation</td></tr>
<tr><td>0</td><td>0</td><td rowspan="2">During EGB synchronization or automatic operation</td></tr>
<tr><td>0</td><td>1</td></tr>
</table>

> **NOTE**
> Parameters ARE and ARO are valid when bit 3 (ART) of parameter No. 7702 is set to 1 (when the retract function executed when an alarm is issued).

7709	Number of the axial feed axis for helical gear compensation

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to Number of controlled axes

This parameter sets the number of the axial feed axis for a helical gear.

> **NOTE**
> When this parameter is set to 0 or a value outside the valid setting range, the Z-axis becomes the axial feed axis.

7710	Controlled axis number for the spindle EGB slave axis

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to Number of controlled axes

This parameters sets the controlled axis number for the spindle EGB slave axis.

> **NOTE**
> 1 Set this parameter when there are two or more groups of servo and spindle EGBs in the same path. Set 0 when there is one group of EGBs in the same path.
> 2 When there are two or more groups of servo and spindle EGBs in the same path, setting a value outside the valid data range in this parameter causes alarm PS1593 to be issued.
> 3 For Series 16*i*, when a value outside the valid data range is set in this parameter, the fourth axis is assumed according to the specifications.
> 4 The setting of this parameter becomes valid after the power is turned off then back on.

	#7	#6	#5	#4	#3	#2	#1	#0
7731	HAD		HBR					EFX

[Input type] Parameter input
[Data type] Bit path

#0 EFX As the EGB command:
0: G80 and G81 are used.
1: G80.4 and G81.4 are used.

> **NOTE**
> When this parameter is set to 0, no canned cycle for drilling can be used.

#5 HBR In EGB synchronization start command G81.4, the number of teeth is:
0: Specified in T.
1: Specified in R.

> **NOTE**
> This parameter is valid when bit 0 (EFX) of parameter No. 7731 is set to 1.

#7 HAD In electronic gear box, the timing for reflecting helical gear compensation and travel distance of automatic phase synchronization to absolute coordinates is:
0: When synchronization is canceled.
1: During helical gear compensation and automatic phase synchronization.

7740	Feedrate during retraction

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the feedrate during retraction for each axis.

7741	Retract amount

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the retract amount for each axis.

7772	Number of position detector pulses per rotation about the tool axis

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[Valid data range] 1 to 999999999
This parameter sets the number of pulses per rotation about the tool axis (master axis), for the position detector.
When the increment system is IS-B, set 360000.

7773	Number of position detector pulses per rotation about the workpiece axis

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[Valid data range] 1 to 999999999
This parameter sets the number of pulses per rotation about the workpiece axis (slave axis), for the position detector.
When the increment system is IS-B, set 360000.

Alarm and message

Number	Message	Description
PS1593	EGB PARAMETER SETTING ERROR	Error in setting a parameter related to the EGB (1) The slave axis specified with G81 is not set as a rotation axis. (bit 0 (ROTx) of parameter No. 1006) (2) Number of pulses per rotation (parameter No. 7772 or 7773) is not set. (3) Although G81 is specified, a parameter such as parameter No. 7710 or 4352 is not set. (4) The slave axis specified in G81 is set as the slave axis of the simple spindle EGB.
PS1594	EGB FORMAT ERROR	Error in the format of the block of an EGB command (1) T (number of teeth) is not specified in the G81 block. (2) In the G81 block, the data specified for one of T, L, P, and Q is out of its valid range. (3) In the G81 block, only one of P and Q is specified.
PS1595	ILL-COMMAND IN EGB MODE	During synchronization with the EGB, a command that must not be issued is issued. (1) Slave axis command using G27, G28, G29, G30, G33, G53, etc. (2) Inch/metric conversion command using G20, G21, etc. (3) The Cs contour control mode is not selected for the slave axis.
PS1596	EGB OVERFLOW	An overflow occurred in the calculation of the synchronization coefficient.

1.9.3 Electronic Gear Box Automatic Phase Synchronization

Overview

In the electronic gear box (EGB), when the start or cancellation of synchronization is specified, the synchronizing state is changed gradually by applying acceleration/deceleration. This is because if synchronization is started or canceled immediately, a shock applies to the machine. Therefore, synchronization can be started or canceled while the spindle is rotating. Also, synchronization ratio can be changed while the spindle is rotating.

In addition, automatic phase synchronization is performed in such a way that, at the start of synchronization, the position of the machine coordinate origin for the workpiece axis matches the spindle position determined by the one-rotation signal. With this synchronization, the same operation is performed as synchronization start caused by a one-rotation signal in hobbing synchronization when using the functions of a hobbing machine.

The spindle corresponds to the EGB master axis and the workpiece axis corresponds to an EGB slave axis (servo axis or Cs contouring axis).

Format

M

- **Acceleration/deceleration type**

G81 T _ L _ R1 ;	Synchronization start
G80 R1 ;	Synchronization cancellation

T : Number of teeth (range of valid settings: 1-5000)
L : Number of hob threads (range of valid settings: -250 to 250, excluding 0)
When L is positive, the direction of rotation about the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation about the workpiece axis is negative (- direction).

- **Acceleration/deceleration plus automatic phase synchronization type**

G81 T _ L _ R2 ;	Synchronization start
G80 R1 ;	Synchronization cancellation

T : Number of teeth (range of valid settings: 1-5000)
L : Number of hob threads (range of valid settings: -250 to 250, excluding 0)
When L is positive, the direction of rotation about the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation about the workpiece axis is negative (- direction).

T

To use this function for the T series, set the following parameters.
Automatic phase synchronization is enabled in a command compatible with that for a hobbing machine used in the T series.
Bit 0 (EFX) of parameter No. 7731=1
Bit 5 (HBR) of parameter No. 7731=1
Bit 6 (PHS) of parameter No. 7702=1

- **Acceleration/deceleration plus automatic phase synchronization type**

G81.4 R _ L _ ;	Synchronization start
G80.4 ;	Synchronization cancellation

R : Number of teeth (range of valid settings: 1-5000)
L : Number of hob threads (range of valid settings: -250 to 250, excluding 0)
When L is positive, the direction of rotation about the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation about the workpiece axis is negative (- direction).

Explanation

- Acceleration/deceleration type

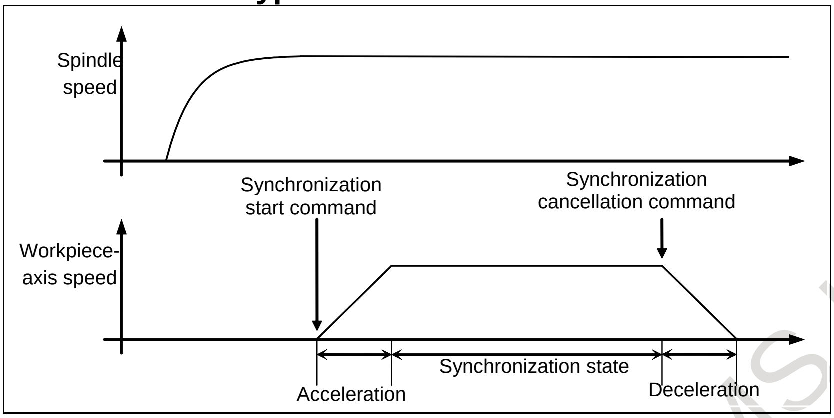


Fig. 1.9.3 (a)

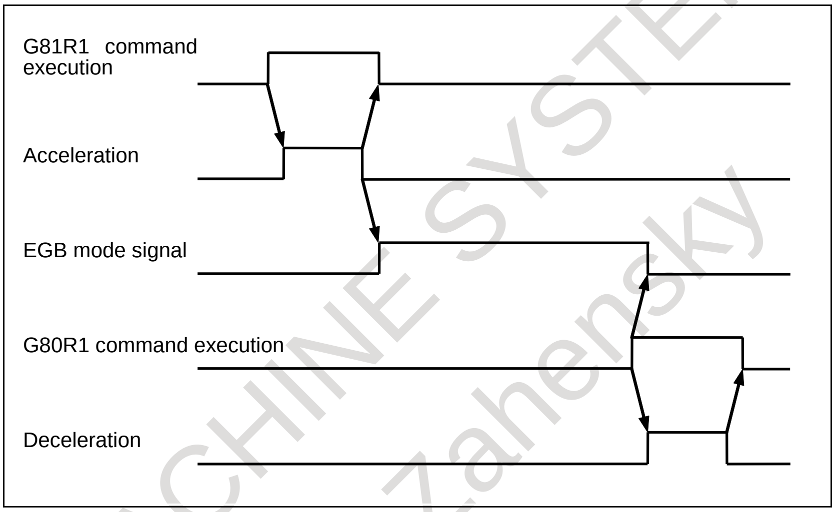


Fig. 1.9.3 (b)

1. Specify G81R1 to start synchronization.
   When G81R1 is specified, the workpiece axis (slave axis) is subject to acceleration at the acceleration rate set in parameter No. 7778. When the synchronization speed is reached, the EGB mode signal, SYNMOD<Fn065.6>, becomes 1 and the G81R1 block is terminated.
2. For cancellation, specify G80R1 while the tool is moved away from the workpiece.
3. When G80R1 is specified, the EGB mode signal SYNMOD<Fn065.6> becomes "0" and deceleration is started immediately at the acceleration rate set in parameter No. 7778.
   When the speed is reduced to 0, the G80R1 block is terminated.

NOTE
1 During synchronization start/cancellation, acceleration/deceleration is linear.
2 In the automatic cancellation of synchronization due to one of the following causes, deceleration is performed and synchronization is canceled:
<1> Reset
<2> PW0000, "POWER MUST BE OFF"
<3> IO alarm
3 If bit 0 (EFX) of parameter No. 7731 is 0, the canned cycle for drilling cannot be used. To use the canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.

- Acceleration/deceleration plus automatic phase synchronization type

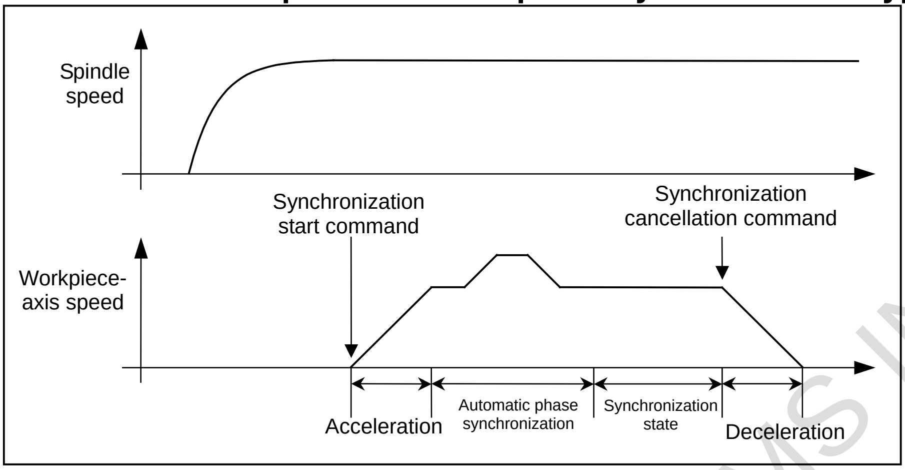


Fig. 1.9.3 (c)

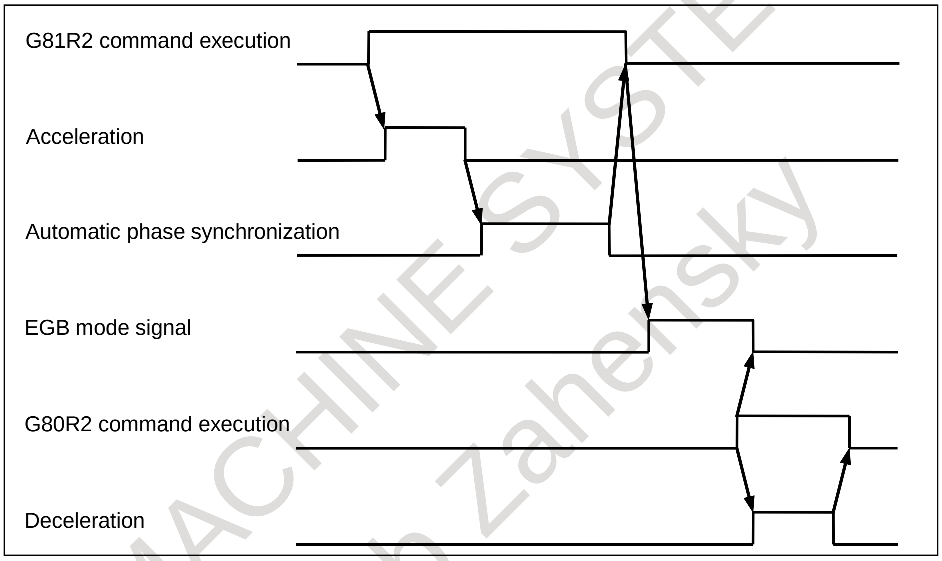


Fig. 1.9.3 (d)

1. Move the workpiece axis to the position that corresponds to that of the one-rotation signal of the spindle.
2. Specify G81R2 to start synchronization.
   When G81R2 is specified, the workpiece axis is accelerated with the acceleration according to the acceleration rate set in the parameter No.7778. Upon completion of phase synchronization, the EGB mode signal SYNMOD<Fn065.6> becomes “1” and the G81R2 block terminates.
3. For cancellation, specify G80R2 while the tool is moved away from the workpiece.
4. When G80R2 is issued, the EGB mode signal SYNMOD<Fn065.6> becomes “0” and deceleration is started immediately according to the acceleration rate set in parameter No. 7778. When the speed reaches 0, the G80R2 block terminates.

⚠ CAUTION

1 Set the automatic phase synchronization speed in parameter No. 7776 and the movement direction in bit 7 (PHD) of parameter No. 7702.

2 Phase synchronization acceleration/deceleration is performed with the rapid traverse linear acceleration/deceleration rate (time constant specified in parameter No. 1620).

⚠ CAUTION

3 The workpiece axis speed is the speed synchronized with spindle rotation with automatic phase synchronization speed being superposed. When setting the position deviation limit parameter No. 1828, take the superposition into consideration.

NOTE

1 The one-rotation signal used for automatic phase synchronization is issued not by the spindle position coder but by the separate Pulsecoder attached to the spindle and used to collect EGB feedback information. This means that the orientation position based on the one-rotation signal issued by the spindle position coder does not match the position used as the reference for the workpiece axis when establishing phase synchronization for automatic phase synchronization based on G81R2.
Moreover, the one-rotation signal of the separate Pulsecoder must be turned on for each rotation of the spindle.

2 With the use of parameter No.7777, the position at which the phase of the workpiece axis is matched can be shifted from the position corresponding to the one-rotation signal in automatic phase matching.

3 Setting bit 6 (EPA) of parameter No. 7731 to 1 can perform automatic phase synchronization in such a way that, at the start of synchronization, the position of the workpiece axis matches the spindle position set by the one-rotation signal.

4 If bit 6 (EPA) of parameter No. 7731 is set to 1 to cause a synchronization command to be issued again when synchronization is already established, automatic phase synchronization is performed to move the workpiece axis in such way that the position where the workpiece axis was when the G81R2 (synchronization command) was issued for the first time matches the spindle position set by the one-rotation signal.

5 In automatic phase synchronization, movement is performed about the workpiece axis from the current position to the nearest phase position in the phase synchronization movement direction specified by the parameter.

6 Linear acceleration/deceleration applies to synchronization start/cancellation.

7 The acceleration/deceleration plus automatic phase synchronization type can be executed by the bit 6 (PHS) of parameter No. 7702 without specifying an R2 command in a G81 or G80 block.

8 In the automatic cancellation of synchronization due to one of the following causes, deceleration is performed and synchronization canceled:
<1> Reset
<2> PW0000, “POWER MUST BE OFF”
<3> IO alarm

9 When EGB (Spindle EGB is excluded) is used, it is necessary to move the separate Pulsecoder attached to the spindle by one rotation or more before executing automatic phase synchronization.

10 When the spindle EGB is used and the master axis is in the speed control mode, perform position coder orientation before executing automatic phase synchronization. In this case, set bit 7 (RFCHK3) of parameter No. 4016 for the master axis to 0 to maintain the spindle one-rotation signal position in the speed control mode.

NOTE

11 When the spindle EGB is used and the master axis is in the Cs contour control mode, perform reference position return before executing automatic phase synchronization.

12 The acceleration rate parameter No.7778 must not be changed in the synchronization mode.

13 If the acceleration rate parameter No. 7778 is 0, alarm PS1598, “EGB AUTO PHASE PARAMETER SETTING ERROR” is issued when G81 is issued.

14 In the Series 16*i*, acceleration for automatic phase matching is set by specifying a feedrate and a time constant in parameters Nos. 2135 and 2136 (Nos. 4384 and 4385 in the case of spindle EGB) separately; in the Series 0*i*-F Plus, acceleration is directly set in parameter No. 7778.

15 If bit 0 (EFX) of parameter No. 7731 is 0, the canned cycle for drilling cannot be used. To use the canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.

16 In the case of bit 7 (HAD) of parameter No.7731 is 0, travel distance of automatic phase synchronization is reflected to absolute coordinates when the synchronization is canceled. In the case of bit 7 (HAD) of parameter No.7731 is 1, travel distance of automatic phase synchronization is reflected to absolute coordinates during automatic phase synchronization.

Direction of rotation (for the spindle EGB)

The EGB automatic phase synchronization function assumes that the direction of rotation about the slave axis is the same as that of rotation about the master axis.

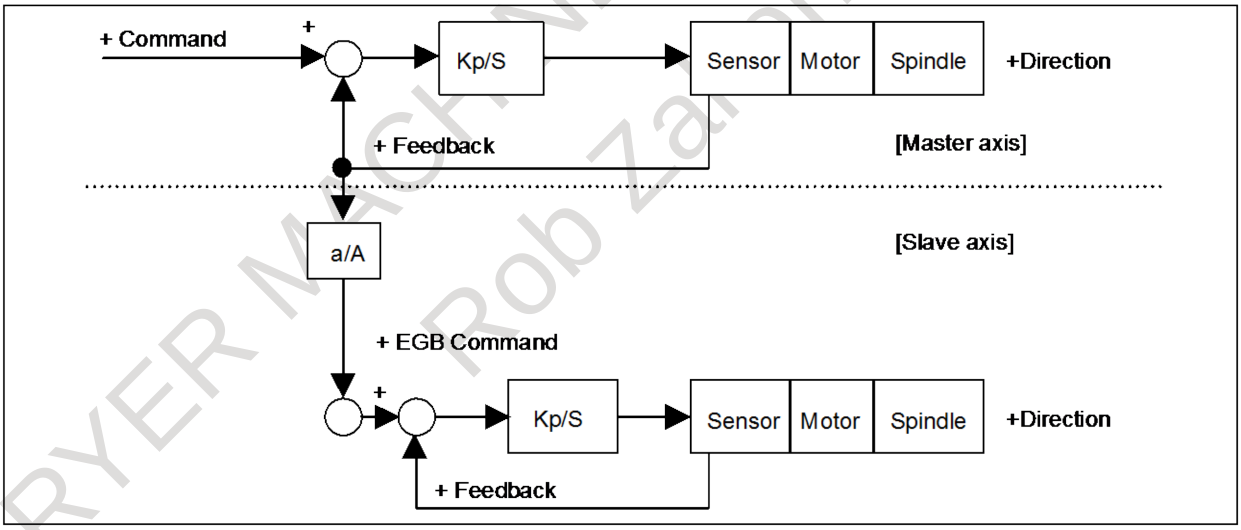


Fig. 1.9.3 (e)

When this function is used, the SFR/SRV function in the Cs contour mode (*) cannot be used.
If you want to change the direction of rotation about the master axis, change the sign of the master axis mode command.

* SFR/SRV function in the Cs contour mode
The SFR/SRV signal determines the direction of rotation of the spindle in the Cs contour mode.

Signal

EGB mode signal SYNMOD< Fn065.6>

[Classification] Output signal
[Function] Reports that synchronization with the EGB is in progress.
[Operation] This signal becomes "1" if:
- Synchronization with the EGB is in progress.

It becomes "0" if:
- Synchronization with the EGB is canceled.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn065		SYNMOD						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
4016	RFCHK3							

[Input type] Parameter input
[Data type] Bit spindle

#6 RFCHK3 When the EGB master axis is in the speed control mode, if you want to execute automatic phase synchronization, be sure to set RFCHK3 for the master axis to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
7702	PHD	PHS						

[Input type] Parameter input
[Data type] Bit path

#6 PHS When the G81/G80 block contains no R command:
0: Acceleration/deceleration is not performed at the start or cancellation of EGB synchronization.
1: Acceleration/deceleration is performed at the start or cancellation of EGB synchronization. After acceleration at the start of synchronization, phase synchronization is automatically performed.

#7 PHD The direction of movement for automatic phase synchronization is:
0: Positive (+).
1: Negative (-).

	#7	#6	#5	#4	#3	#2	#1	#0
7731	HAD	EPA	HBR					EFX

[Input type] Parameter input
[Data type] Bit path

#0 EFX As the EGB command:
0: G80 and G81 are used.
1: G80.4 and G81.4 are used.

NOTE
When this parameter is set to 0, no canned cycle for drilling can be used.

#5 HBR In EGB synchronization start command G81.4, the number of teeth is:
0: Specified in T.
1: Specified in R.

> **NOTE**
> This parameter is valid when bit 0 (EFX) of parameter No. 7731 is set to 1.

#6 EPA Automatic phase synchronization for the electronic gear box is performed in such a way that:
0: The machine coordinate 0 of the slave axis is aligned to the position of the master axis one-rotation signal.
1: The position of the slave axis at synchronization start is aligned to the position of the master axis one-rotation signal. (Specification of Series 16*i*)

#7 HAD In electronic gear box, the timing for reflecting helical gear compensation and travel distance of automatic phase synchronization to absolute coordinates is:
0: When synchronization is canceled.
1: During helical gear compensation and automatic phase synchronization.

7776	Feedrate during automatic phase synchronization for the workpiece axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg/min
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to standard parameter setting table (C).
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the feedrate during automatic phase synchronization for the workpiece axis.
When this parameter is set to 0, the rapid traverse rate (parameter No. 1420) is used as the feedrate during automatic phase synchronization.

7777	Angle shifted from the spindle position (one-rotation signal position) the workpiece axis uses as the reference of phase synchronization

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0.000 to 360.000 (when the increment system is IS-B)
This parameter sets the angle shifted from the spindle position (one-rotation signal position) the workpiece axis uses as the reference of phase synchronization.

7778	Acceleration for acceleration/deceleration for the workpiece axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg/sec^2
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(For a millimeter machine, 0.0 to +100000.0, for an inch machine, 0.0 to +10000.0)
This parameter sets an acceleration for acceleration/deceleration for the workpiece axis.

> **NOTE**
> 1 In the Series 16*i*, acceleration for automatic phase matching is set by specifying a feedrate and a time constant in parameters Nos. 2135 and 2136 (Nos. 4384 and 4385 in the case of spindle EGB) separately; in the Series 0*i*-F Plus, acceleration is directly set in parameter No. 7778.
> 2 If this parameter is 0, alarm PS1598, “EGB AUTO PHASE PARAMETER SETTING ERROR” is issued when G81 is issued.

Alarm and message

Number	Message	Description
PS1597	EGB AUTO PHASE FORMAT ERROR	Format error in the G80 or G81 block in EGB automatic phase synchronization (1) R is outside the permissible range. (2) In the spindle EGB, reference position return is not performed for the master axis before G81R2 is specified.
PS1598	EGB AUTO PHASE PARAMETER SETTING ERROR	Error in the setting of a parameter related to EGB automatic phase synchronization (1) The acceleration/deceleration parameter is not correct. (2) The automatic phase synchronization parameter is not correct.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Electronic gear box automatic phase synchronization

1.9.3.1 Arbitrary Gear Ratio for Master Axis in Electronic Gear Box Automatic Phase Synchronization

Overview

In electronic gear box automatic phase synchronization, the gear ratio between the master axis and the detector must be 1-to-1. Because the angle of the master axis is counted by number of pulses from one-rotation signal of detector.
This function considers the pulse from the detector and the gear ratio for calculating the angle of master axis. Therefore, EGB automatic phase synchronization can apply even if the gear ratio is not 1-to-1.
Moreover, the angle about the master axis can be reset by angle counter reset signal PREGB<Gn291.4>. An arbitrary angle of the master axis can be specified as the reference position of phase synchronization by this signal.

Explanation

1. To enable this function set bit 5 (EGEx) of parameter No.2273 to 1.
2. The value of counter per rotation for the master axis is set by 2 parameters. The parameters mean the fraction. Numerator is set to parameter No.1821, and denominator is set to parameter No.2179.
   For example, when the number of pulses from detector is 4096 and the gear ratio of the master axis to the detector is 1 : 7, the setting is as follows.
   No.1821 = 4096
   No.2179 = 7
3. To reset the angle about the master axis, set the angle counter reset signal PREGB<Gn291.4> to “1”. When angle counter reset signal PREGB is set to “1”, CNC resets the angle about the master axis. At this time, angle counter reset completion signal PREGBF<Fn291.4> is set to “1”.

- When the bit 6 (EPA) of parameter No.7731 = 0:
  In EGB automatic phase synchronization, the machine coordinate 0 for the slave axis is aligned to the position of angle 0 for the master axis.
- When the bit 6 (EPA) of parameter No.7731 = 1:
  In EGB automatic phase synchronization, the position for the slave axis at synchronization start is aligned to the position of angle 0 for the master axis.

NOTE

1. This function is the additional specification in EGB automatic phase synchronization for servo axis.
2. This function cannot be used in EGB automatic phase synchronization for spindle EGB.
3. If the serial type position detector , or A/B phase type position detector(FFG setting other than 1/1) are used in EGB automatic phase synchronization, this function must be made effective.
4. When bit 5 (EGEx) of parameter No.2273 is set to 0, parameters No.1821, No.2179, angle counter reset signal PREGB<Gn291.4> and angle counter reset completion signal PREGBF<Fn291.4> are invalid.
5. This function cannot be used in EGB (FSSB type)

Signal

Angle counter reset signal PREGB<Gn291.4>

[Classification] Input signal
[Function] Reset the angle about the master axis.
[Operation] When this signal is set to “1”, the angle about the master axis is reset.

Angle counter reset completion signal PREGBF<Fn291.4>

[Classification] Output signal
[Function] This signal indicates that the reset of angle about the master axis is completed.
[Operation] This signal becomes 1 under the following condition.
- Reset of the angle is completed.

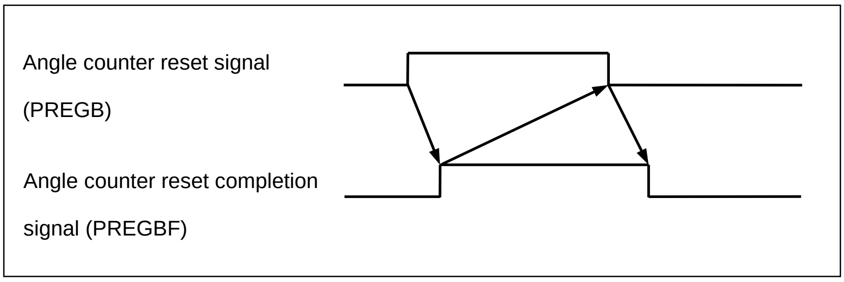
This signal becomes 0 under the following condition.
- Angle counter reset signal PREGB<Gn291.4> is set to “0”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn291				PREGB				

	#7	#6	#5	#4	#3	#2	#1	#0
Fn291				PREGBF				

Timing chart



Parameter

Set the following parameters for the EGB dummy axis.

	#7	#6	#5	#4	#3	#2	#1	#0
2273		EGFx	EGEx					

[Input type] Parameter input
[Data type] Bit axis

#5 EGEx In electronic gear box automatic phase synchronization, the gear ratio of the master axis to the detector is:
0: 1-to-1.
1: Arbitrary.

#6 EGFx FFG is:
0: Not considered in the number of pulses per rotation about the master axis, for the position detector.
1: Considered in the number of pulses per rotation about the master axis, for the position detector.

> **NOTE**
> This parameter is valid for A/B phase type position detector. In the case of using serial type position detector, FFG is always considered.

In case of considering FFG, set the value which FFG is multiplied by the number of output pulses of detector to parameter No.7773. In case of bit 5 (EGEx) of parameter No.2273 is set to 1, set the value in which FFG is multiplied to the counter per rotation for the master axis.
N: Numerator of FFG (parameter No.2084)
M: Denominator of FFG (parameter No.2085)
α: Counter per rotation for the master axis (parameter No.1821 / parameter No.2179)
The value of new counter is the old one multiplied by FFG.

$$\alpha\,[\text{New}] = \alpha\,[\text{Old}] \times \frac{N}{M}$$

2084	Flexible feed gear (numerator) (N)

2085	Flexible feed gear (denominator) (M)

[Input type] Parameter input
[Data type] Word axis
In the case of serial type position detector or A/B phase type position detector and bit 6 (EGFx) of parameter No.2273 is set to 1, FFG is considered to the synchronization coefficient.

> **NOTE**
> If FFG is not using, be sure to set to 1 for both numerator and denominator parameter.

1821	Counter per rotation for the master axis (numerator)

[Input type] Parameter input
[Data type] 2-Word axis
[Valid data range] 1 to 2147483647

Set the counter per one rotation for the master axis with a fraction. (Numerator)

NOTE
When 0 or less is set to this parameter, the alarm PS1598, "EGB AUTO PHASE PARAMETER SETTING ERROR" occurs when G81 is issued.

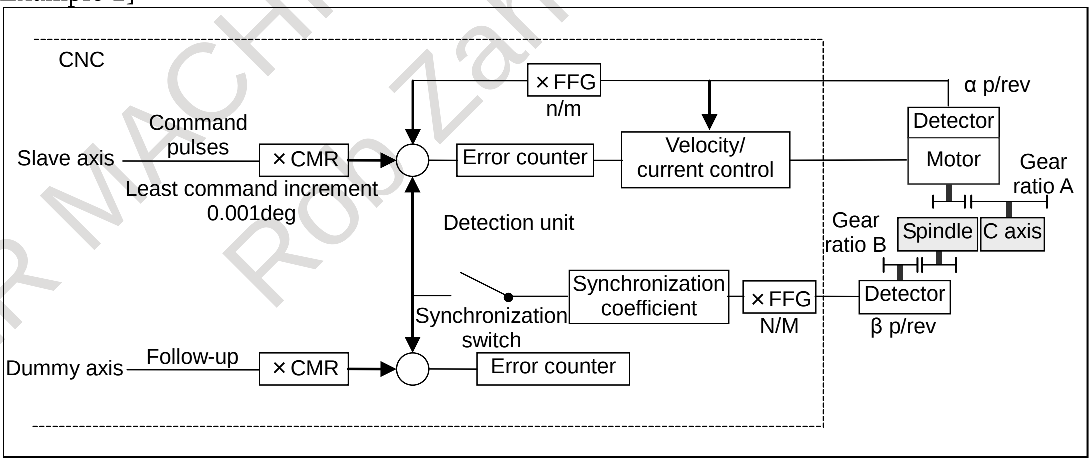
2179	Counter per rotation for the master axis (denominator)

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 1 to 32767

Set the counter per one rotation for the master axis with a fraction. (Denominator)

NOTE
1 When 0 is set to this parameter, the alarm PS1598, "EGB AUTO PHASE PARAMETER SETTING ERROR" occurs when G81 is issued.
2 When negative value is set to this parameter, the alarm SV0417 "ILL DGTL SERVO PARAMETER" occurs.

[Example 1]



Gear ratio of the spindle to the detector B:
2/3 (When the detector rotates 2 for 3 spindle rotations)
Number of detector pulses per spindle rotation β:
80,000 pulses/rev (Calculated as 4 pulses for one A/B phase cycle)
FFG N/M of the EGB dummy axis : 1/1
Gear ratio of the C-axis A:
1/36 (One C-axis rotation corresponds to 36 motor rotations)
Number of detector pulses per C-axis rotation α:
1,000,000 pulses/rev
C-axis CMR : 1
C-axis FFG n/m : 1/100

Because the gear ratio of the spindle (master axis) to the detector is 2/3, enable this function by setting 1 to bit 5 (EGEx) of parameter No.2273.
The gear ratio of the spindle to the detector is 2/3 and the detector is 80,000 pulse/rev, so the angle counter is:

$$80000 \times \frac{2}{3} = \frac{160000}{3}$$

Set the parameter No.1821=160,000 and parameter No.2179=3.
The angle counter is not effective to synchronization ratio. The ordinary parameter No.7772 is used for calculation of synchronization ratio.
The number of pulses per spindle one-rotation is:

$$80000 \times \frac{2}{3} = \frac{160000}{3} \qquad <1>$$

It cannot be divided. It is impossible to set. In this case, the setting of the parameter No.7773 must be changed, and the ratio of the parameter No.7772 and the parameter No.7773 must be changed to the ratio that wants to be set.

Number of pulses per C-axis rotation by detection unit is:

$$1000000 \div 1/36 \times 1/100 = 360000 \qquad <2>$$

For <1>,<2>

$$\frac{\text{No.7772}}{\text{No.7773}} = \frac{160000/3}{360000} = \frac{160000}{360000 \times 3} = \frac{160000}{1080000}$$

Therefore, it is set as parameter No.7772 = 160,000 and parameter No.7773 = 1,080,000.
As described above, all the setting of parameter No.7772 and parameter No.7773 have to do is to indicate the ratio correctly. So, you can reduce the fraction indicated by the settings. For example, you may set 16 for parameter No.7772 and 108 for parameter No.7773 for this case.

[Example 2]
When the FFG of EGB dummy axis N/M is 1/10 for the above example, bit 6 (EGFx) of parameter No.2273 is set to 1 for making valid FFG.

The parameter No.7772 and parameter No.7773 is set to the new value that multiplies FFG for considering FFG.

$$\frac{\text{No.7772}}{\text{No.7773}} = \frac{160000}{1080000} \times \frac{1}{10} = \frac{16000}{1080000}$$

Set the parameter No.7772 = 16,000 and parameter No.7773 = 1,080,000.

And a new value that the angle counter similarly multiplied FFG is set.

$$\frac{\text{No.1821}}{\text{No.2179}} = \frac{160000}{3} \times \frac{1}{10} = \frac{16000}{3}$$

Set the parameter No.1821 = 16,000 and parameter No.2179 = 3.

Alarm and message

Number	Message	Description
PS1598	EGB AUTO PHASE PARAMETER SETTING ERROR	Error in the setting of a parameter related to EGB automatic phase synchronization (1) The acceleration/deceleration parameter is not correct. (2) The automatic phase synchronization parameter is not correct. (3) Bit 5 (EGE) of parameter No.2273 is set to 1 with the servo software that does not support this function,.
SV0417	ILL DGTL SERVO PARAMETER	A digital servo parameter setting is incorrect.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	Electronic Gear Box Automatic Phase Synchronization

1.9.3.2 Speed-up EGB Automatic Phase Synchronization

Overview

In electronic gear box automatic phase synchronization, the next block can be started without waiting for completion of phase synchronization.
It is possible to set M code for waiting for completion of EGB automatic phase synchronization in parameter No.7787. When the M code for waiting for completion of EGB automatic phase synchronization is commanded, execution of the next block is started after completion of EGB automatic phase synchronization.
This makes it possible to issue a program command during auto phase synchronization and acceleration of auto phase synchronization.
Also, by commanding the M code for waiting for completion of EGB automatic phase synchronization before the start of machining, it is possible to start machining with the EGB automatic phase synchronization completed.
It is possible to set M code for starting deceleration of EGB automatic phase synchronization in parameter No.7788. When the M code for starting deceleration of EGB automatic phase synchronization is commanded, deceleration of EGB automatic phase synchronization is started. The block of M code for starting deceleration of EGB automatic phase synchronization is immediately ended and the next block is started.
This makes it possible to issue a program command during deceleration of EGB automatic phase synchronization.

Format

- **Acceleration/deceleration plus speed-up automatic phase synchronization type**

G81 T_ L_ R3 ;	**Synchronization start**
G80 R3 ;	**Synchronization cancellation**

T : Number of teeth (range of valid settings: 1-5000)
L : Number of hob threads (range of valid settings: -250 to 250, excluding 0)
When L is positive, the direction of rotation about the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation about the workpiece axis is negative (- direction).

Explanation

- Acceleration/deceleration plus speed-up automatic phase synchronization type

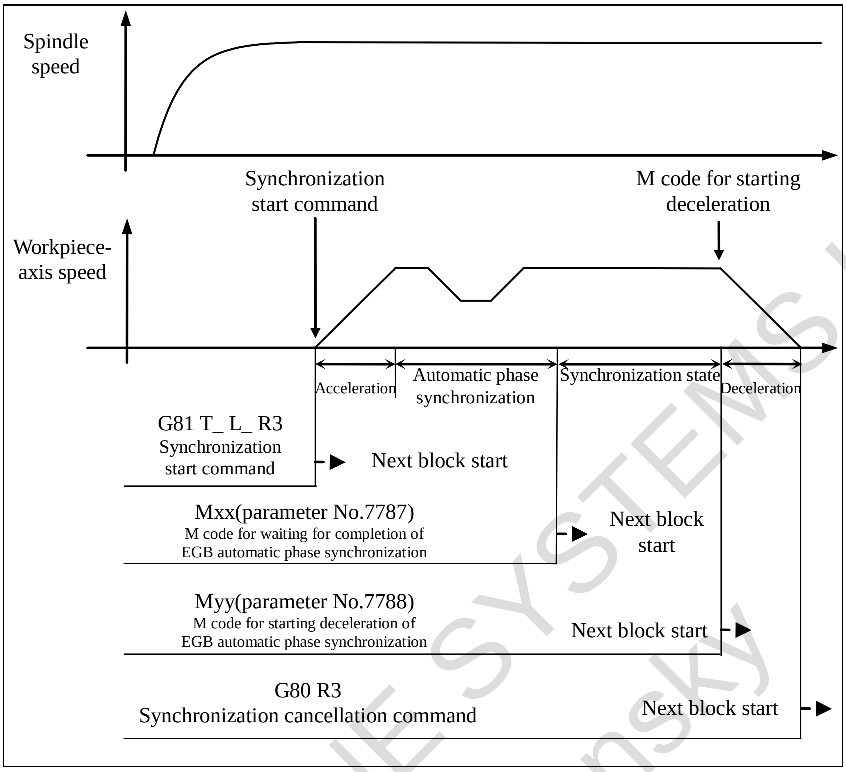


Fig. 1.9.3.2 (a) Acceleration/deceleration plus speed-up automatic phase synchronization type

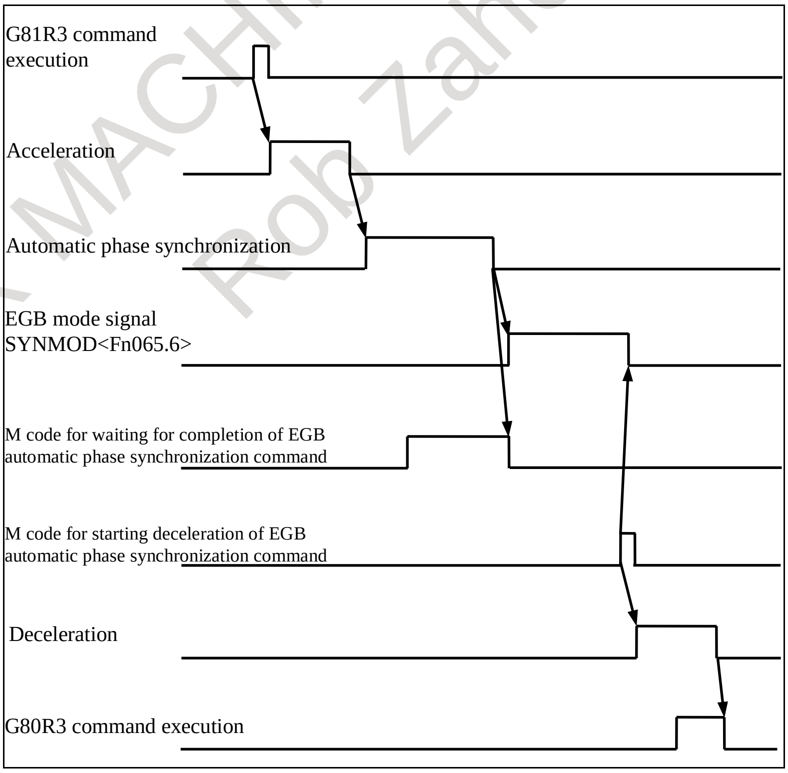


Fig. 1.9.3.2 (b) Timing chart of acceleration/deceleration plus speed-up automatic phase synchronization type

1. Move the workpiece axis to the position that corresponds to that of the one-rotation signal of the spindle.

2. Specify G81R3 to start synchronization. The block of G81R3 is immediately ended and the next block is started.
   When G81R3 is specified, the workpiece axis is accelerated with the acceleration according to the acceleration rate set in the parameter No.7778. Upon completion of phase synchronization, the EGB mode signal SYNMOD<Fn065.6> becomes “1”.
3. When the M code for waiting for completion of EGB automatic phase synchronization is commanded, the next block is not started until automatic phase synchronization is completed.
4. For cancellation, specify the M code for starting deceleration of EGB automatic phase synchronization while the tool is moved away from the workpiece. The block of M code for starting deceleration is immediately ended and the next block is started.
5. When M code for starting deceleration is issued, the EGB mode signal SYNMOD<Fn065.6> becomes “0” and deceleration is started immediately according to the acceleration rate set in parameter No.7778.
6. When G80R3 is issued, deceleration is continued. And the speed reaches 0, the G80R3 block terminates.

NOTE

1 Do not specify EGB synchronization (G81/G81.5) command before automatic phase synchronization is completed. If EGB synchronization command is issued, alarm (PS1595), “ILL-COMMAND IN EGB MODE” is generated. Even when an alarm (PS1595) occurs, automatic phase synchronization before the occurrence of an alarm is continued.

2 Confirm the completion of phase synchronization by M code (parameter No.7787) for waiting for completion of EGB automatic phase synchronization, then issue a movement command or machining to work axis. Also, in the case of helical gears, make sure that the phase synchronization is completed and then issue a movement command for the axial feed axis.

3 Specify M code (parameter No.7787) for waiting for completion of EGB automatic phase synchronization and M code (parameter No.7788) for starting deceleration of EGB automatic phase synchronization in a single block.

4 As for M code (parameter No.7787) for waiting for completion of EGB automatic phase synchronization and M code (parameter No.7788) for starting deceleration of EGB automatic phase synchronization, the function code signal and strobe signal are not output.

- Program example

```
O1000 ;
  :
G81 T_ L_ R3;   Synchronization start command
G00 X_;         X-axis approach
Mxx;            M code for waiting for completion of EGB automatic phase synchronization command
G01 Z_ F_;      Machining start
  :
G00 X_;         Release work and tool
Myy;            M code for starting deceleration of EGB automatic phase synchronization command
G00 X_;         X-axis retraction
G80 R3;         Synchronization cancellation command
  :
M30;
```

Signal

EGB mode signal SYNMOD< Fn065.6>

[Classification] Output signal

[Function] Reports that synchronization with the EGB is in progress.

[Operation] This signal becomes “1” if:

- Synchronization with the EGB is in progress.

It becomes “0” if:

- Synchronization with the EGB is canceled.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn065		SYNMOD						

Parameter

7787	M code for waiting for completion of EGB automatic phase synchronization

[Input type] Parameter input

[Data type] 2-word path

[Valid data range] 0 to 99999999

Set M code for waiting for completion of EGB automatic phase synchronization.
When 0 is set in this parameter, this parameter is invalid.

7788	M code for starting deceleration of EGB automatic phase synchronization

[Input type] Parameter input

[Data type] 2-word path

[Valid data range] 0 to 99999999

Set M code for starting deceleration of EGB automatic phase synchronization.
When 0 is set in this parameter, this parameter is invalid.

1.9.4 Skip Function for EGB Axis

Overview

This function enables the skip or high-speed skip signal (these signals are collectively called skip signals in the remainder of this manual) for the EBG slave axis in synchronization mode with the EGB (electronic gear box).

This function has features such as the following:

1 If a skip signal is input while an EGB axis skip command block is being executed, this block does not terminate until the specified number of skip signals have been input.

2 If a skip signal is input while an EGB axis skip command block is being executed, the tool remains in synchronization mode and moves, not stopping on the EGB slave axis.

3 The machine coordinates assumed when skip signals are input and the number of input skip signals are stored in specified custom macro variables.

For an explanation of the electronic gear box, see the preceding Subsection, "Electronic Gear Box" in this manual.

Format

G81 T_ L_ ;	**EGB mode ON**
G31.8 G91 α0 P_ Q_ (R_) ;	**EGB skip command**

α: Specify an EGB slave axis. The specified value must always be 0.
P: Number of the first one of the custom macro variables used to store the machine coordinates assumed when skip signals are input.
Q: Number of skip signals that can be input during the execution of G31.8 (permissible range: 1 to 512).
R: Number of the custom macro variable used to store the number of input skip signals.
Specify it to check the number of input signals.

Explanation

G31.8 is a one-shot G code.
When G31.8 is executed, the machine coordinates assumed when skip signals are input are written in as many custom macro variables as the number specified in Q, starting with the one having the number specified in P, when the skip command block terminates.
Also, the number of input skip signals is written to the custom macro variable specified in R each time a skip signal is input.

Positional deviation

In skip function for EGB axis, when bit 5 (ESE) of parameter No.6216 is set to 1, positional deviation when the skip signal and the high-speed skip signal are input is considered and compensated.
As a result, the measurement with high accuracy becomes possible in skip function for EGB axis.

Example

The pitch of a gear can be measured.

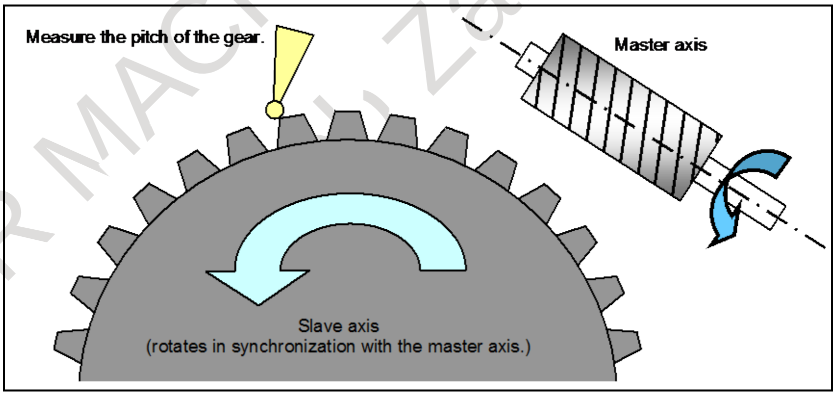


Fig. 1.9.4

G81 T200 L2 ; ....EGB mode ON
X_ ;
Z_ ;
G31.8 G91 C0 P500 Q200 R1 ;...........EGB skip command

After 200 skip signals have been input, the 200 skip positions on the C-axis that correspond to the respective skip signals are stored in custom macro variables #500 to #699.
Also, the number of input skip signals is stored in custom macro variable #1.

NOTE

1. When specifying this function, specify only a single EGB slave axis. If no axis is specified for two or more axes are specified, alarm PS1152, “G31.9/G31.8 FORMAT ERROR” is generated.
2. If P is not specified, alarm PS1152 is generated.
3. If R is not specified, the number of input skip signals is not written to a custom macro variable.
4. The custom macro variable numbers specified in P and R must be existing ones. If a non-existent variable number is specified, alarm PS0115, "VARIABLE NO. OUT OF RANGE" is generated.
   If a variable shortage occurs, alarm PS0115 is generated.
5. Whether to use conventional skip signals or high-speed skip signals with this function can be specified with HSS, bit 4 of parameter No. 6200. If deciding to use high-speed skip signals, specify which high-speed signals to enable with 9S1 to 9S8, bits 0 to 7 of parameter No. 6208).
6. Skip positions are calculated from feedback pulses from the machine. Thus, errors due to delay in acceleration/deceleration and the servo system are small. In skip function for EGB axis, when bit 5 (ESE) of parameter No.6216 is set to 1, positional deviation when the skip signal and the high-speed skip signal are input is considered and compensated.

Signal

For details of skip signals, see the sections on "skip function" and "high-speed signal".

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200		SRE	SLS	HSS			SK0	GSK

[Input type] Parameter input
[Data type] Bit path

#0 GSK As a skip signal, the skip signal SKIPP is:
0: Invalid.
1: Valid.

#1 SK0 This parameter specifies whether the skip signal is made valid under the state of the skip signal SKIP and the multistage skip signals SKIP2 to SKIP8.
0: Skip signal is valid when these signals are “1”.
1: Skip signal is valid when these signals are “0”.

#4 HSS
0: The skip function does not use high-speed skip signals while skip signals are input. (The conventional skip signal is used.)
1: The step skip function uses high-speed skip signals while skip signals are input.

#5 SLS
0: The multi-step skip function does not use high-speed skip signals while skip signals are input. (The conventional skip signal is used.)
1: The multi-step skip function uses high-speed skip signals while skip signals are input.

> **NOTE**
> The skip signals (SKIP and SKIP2 to SKIP8) are valid regardless of the setting of this parameter. They can also be disabled using bit 4 (IGX) of parameter No. 6201.

#6 SRE When a high-speed skip signal or high-speed measurement position arrival signal is used:
0: The signal is assumed to be input on the rising edge (contact open → close).
1: The signal is assumed to be input on the falling edge (contact close → open).

	#7	#6	#5	#4	#3	#2	#1	#0
6201	**SKPXE**		**CSE**	**IGX**				

[Input type] Parameter input
[Data type] Bit path

#4 IGX When the high-speed skip function is used, SKIP, SKIPP, and SKIP2 to SKIP8 are:
0: Enabled as skip signals.
1: Disabled as skip signals.

#5 CSE For the continuous high-speed skip command, high-speed skip signals are:
0: Effective at either a rising or falling edge (depending on the setting of bit 6 (SRE) of parameter No. 6200).
1: Effective at both the rising and falling edges.

#7 SKPXE For the skip function (G31), the skip signal SKIP is:
0: Enabled.
1: Disabled.

Whether the skip signals are enabled or disabled

Parameter	Bit 4 (IGX) of parameter No. 6201	Bit 0 (GSK) of parameter No. 6200	Bit 7 (SKPXE) of parameter No. 6201	Skip signal SKIPP	Skip signal SKIP	Multistage skip signals SKIP2-SKIP8
Setting	0	0	0	Disabled	Enabled	Enabled
	0	1	0	Enabled	Enabled	Enabled
	0	0	1	Disabled	Disabled	Enabled
	0	1	1	Enabled	Disabled	Enabled
	1	0	0	Disabled	Disabled	Disabled
	1	1	0	Disabled	Disabled	Disabled
	1	0	1	Disabled	Disabled	Disabled
	1	1	1	Disabled	Disabled	Disabled

Bit 4 (IGX) of parameter No. 6201 is valid for the skip function using high-speed skip signals (when bit 4 (HSS) of parameter No. 6200 is set to 1) or for the multistage skip function using high-speed skip signals (when bit 5 (SLS) of parameter No. 6200 is set to 1).

	#7	#6	#5	#4	#3	#2	#1	#0
6208	**9S8**	**9S7**	**9S6**	**9S5**	**9S4**	**9S3**	**9S2**	**9S1**

[Input type] Parameter input
[Data type] Bit path

9S1 to 9S8 Specify which high-speed skip signal is enabled for the continuous high-speed skip command G31P90 or the EGB skip command G31.8.

The settings of each bit have the following meaning:
0: The high-speed skip signal corresponding to the bit is disabled.
1: The high-speed skip signal corresponding to the bit is enabled.
The bits correspond to signals as follows:

Parameter	High-speed skip signal	Parameter	High-speed skip signal
9S1	HDI0	9S5	HDI4
9S2	HDI1	9S6	HDI5
9S3	HDI2	9S7	HDI6
9S4	HDI3	9S8	HDI7

	#7	#6	#5	#4	#3	#2	#1	#0
6216			ESE					

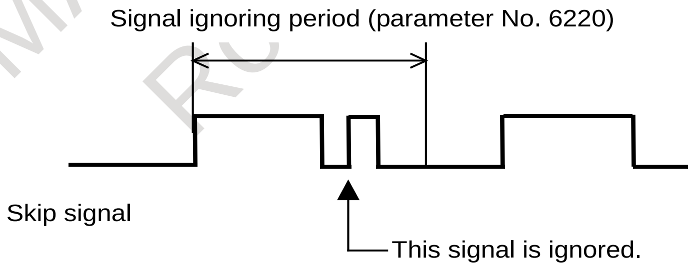
[Input type] Parameter input
[Data type] Bit path

#5 ESE positional deviation when the skip signal and the high-speed skip signal are input:
0: is not considered.
1: is considered.

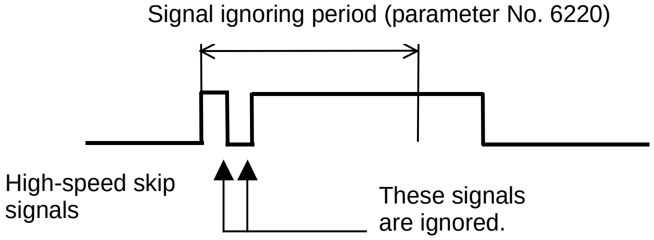
6220	**Period during which skip signal input is ignored for the continuous high-speed skip function and EGB axis skip function**

[Input type] Parameter input
[Data type] Byte path
[Unit of data] 8msec
[Valid data range] 3 to 127(× 8msec)

This parameter specifies the period from when a skip signal is input to when the next skip signal can be input for the continuous high-speed skip function and EGB axis skip function. This parameter is used to ignore chattering in skip signals.
If a value that falls outside the valid range is specified, the setting is assumed to be 24 msec.



When high-speed skip signals are used and bit 5 (CSE) of parameter No. 6201 is set to 1, signals are handled as follows:



Alarm and message

Number	Message	Description
PS0115	VARIABLE NO. OUT OF RANGE	A number that cannot be used for a local variable, common variable, or system variable in a custom macro is specified. In the EGB axis skip function (G31.8), a non-existent custom macro variable number is specified. Or, the number of custom macro variables used to store skip positions is not sufficient.
PS1152	G31.9/G31.8 FORMAT ERROR	There is a format error in the G31.9 or G31.8 block as described below. - No axis address is specified in the G31.9 or G31.8 block. - More than axis address is specified in the G31.9 or G31.8 block. - P is not specified in the G31.9 or G31.8 block

1.9.5 Electronic Gear Box 2 Pair

Overview

The Electronic Gear Box is a function for rotating a workpiece in sync with a rotating tool, or to move a tool in sync with a rotating workpiece. With this function, the high-precision machining of gears, threads, and the like can be implemented. A desired synchronization ratio can be programmed.
Up to two sets of axes can be synchronized. A gear grinding machine can be controlled, for instance, by using one axis for rotating the workpiece in sync with the tool and another axis for performing dressing in sync with the tool.
The electronic gear box is hereinafter called an EGB function.

1.9.5.1 Specification method (G80.5, G81.5)

Format

G81.5 $\left\{\begin{array}{l}\mathbf{T\,t}\\ \mathbf{P\,p}\end{array}\right\}$ $\left\{\begin{array}{l}\beta\,\mathbf{j}\\ \beta\,\mathbf{0}\ \ \mathbf{L}\,l\end{array}\right\}$ **;** Synchronization start

↑ Amount of travel along the master axis

↑ Amount of travel along the slave axis

G80.5 β0 ; Synchronization cancellation

Explanation

- Master axis, slave axis, and dummy axis

The synchronization reference axis is called the master axis, while the axis along which movement is performed in synchronization with the master axis is called the slave axis. For example, if the workpiece moves in synchronization with the rotating tool as in a hobbing machine, the tool axis is the master axis and the workpiece axis is the slave axis.
Which axes to become the master and slave axes depends on the configuration of the machine. For details, refer to the manual issued by the machine tool builder.
A single servo axis is used exclusively so that digital servo can directly read the rotation position of the master axis. (This axis is called the EGB dummy axis.)

- Synchronization start

When the ratio of the master-axis travel to the slave-axis travel is specified, synchronization starts.

<u>Specify the master-axis travel in either of the following ways.</u>

1 Master-axis speed

T t: Master-axis speed (1≤ t ≤5000)

2 Master-axis pulse count

P p : Master-axis pulse count ($1 \leq p \leq 999999999$)

Specify a pulse count on the condition that four pulses correspond to one period in the A and B phases.

Specify the slave-axis travel in either of the following ways.

1 Slave-axis travel

βj : Slave-axis address

j : Slave-axis travel indicated in units of the minimum travel increments(the range of valid settings for usual axis movement applies)

When j = 0, the specified command is regarded as being a command for the slave-axis speed, described below. In this case, if L is not specified, an alarm is output.

2 Slave-axis speed

β0 L±*l*

β : Slave-axis address

l : Slave axis speed($-250 \leq l \leq 250$, $l = 0$ is excluded, however.)

- Synchronization cancellation

1 Canceling synchronization for each axis by issuing a command

With a G80.5 β0 command, synchronization is canceled.

β is the address of the slave axis. Synchronization of the slave axis specified by β is canceled.

A cancellation command can be issued only for one axis in one block.

When β0 is not specified, the synchronization of all currently synchronized axes is canceled.

When a synchronization cancellation command is issued, the absolute coordinates for the slave axis are updated according to the amount of travel during synchronization. For a rotation axis, the value obtained by rounding off the amount of travel during synchronization to the nearest 360 degrees is added to the absolute coordinates.

2 Canceling synchronization by a reset

By setting HBR, bit 0 of parameter No. 7700, to 0, synchronization is canceled with a reset.

3 Others

Synchronization is automatically canceled under the following conditions.

<1> Emergency stop

<2> Servo alarm

<3> Alarm PW0000 (indicating that the power should be turned off)

<4> An IO alarm is generated

- How to reduce the synchronous error

When you use the Electronic gear box function, to reduce the synchronous error, please apply feed-forward to the slave axis and set 100% to the parameter of feed-forward coefficient.

And please confirm the effectiveness of feed-forward by the following procedure.

[Procedure]

1. When the slave axis synchronizes only with the command from master axis (ie. When the slave axis doesn’t use helical gear compensation), the position error of slave axis is regarded as the synchronous error. Please check that the position error (DGN data No.300) of the slave axis becomes 0 or so.
2. And also check the position error is near 0 even when the speed of the master axis is changed.

Set the following parameters to use Feed-forward function with 100% coefficient.

[Setting parameters]

Bit 3 (PIENx) of parameter No. 2003 = 1 (Slave axis)	Use PI control in velocity control
Bit 1 (FEEDx) of parameter No. 2005 = 1 (Slave axis)	Use Feed-forward function

Bit 1 (FFALx) of parameter No. 2011 = 1 (Slave axis) Use Feed-forward function irrespective of feed mode
Parameter No.2068 (FF coefficient) = 10000 (Slave axis) Feed-forward coefficient is 100%.

Please refer to the chapter of “Feed-forward Function” in FANUC AC SERVO MOTOR α*i* series FANUC AC SERVO MOTOR β*i* series FANUC LINEAR MOTOR L*i*S series FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR D*i*S series Parameter manual (B-65270EN) about the detail of Feed-forward function.

- How to reduce shock at the start of acc./dec.

If the shock of slave axis is large when the master axis accelerates or decelerates in velocity control mode, please apply “Soft start/stop” function to the master axis (spindle axis).
Please set the following parameters to use Soft start/stop function.

[Setting parameters]

Bit 2 (SOSALWs) of parameter No. 4399 = 1 Use Soft start/stop function even at emergency stop
Note) If the spindle axis is a sub axis of spindle switching control, please set bit 2 of parameter No. 4472 instead of bit 2 of parameter No. 4399.
Parameter No. 4030 Soft start/stop setting time
Parameter No. 4508 Rate of change in acceleration at soft start/stop
Note) Parameters Nos. 4030 and 4508 should be tuned according to the spindle characteristic to reduce the shock well.

[Signals]

Soft start/stop signal SOCNA <Gn071.4> : First spindle
Soft start/stop signal SOCNB <Gn075.4> : Second spindle
Soft start/stop signal SOCNC <Gn205.4> : Third spindle
Soft start/stop signal SOCND <Gn267.4> : Fourth spindle

Please refer to FANUC AC SPINDLE MOTOR α*i*/β*i* series BUILT-IN SPINDLE MOTOR B*i* series parameter manual (B-65280EN) about the detail of Soft start/stop function.

⚠ CAUTION

1. Feed hold, interlock, and machine lock are invalid to a slave axis in EGB synchronization.
2. Even if an OT alarm is issued for a slave axis in EGB synchronization, synchronization will not be canceled.
3. During synchronization, it is possible to execute a move command for a slave axis and other axes, using a program. The move command for a slave axis must be an incremental one.

NOTE

1. If bit 0 (HBR) of parameter No. 7700 is set to 1, EGB synchronization will not be canceled due to a reset. Usually, set this parameter bit to 1.
2. In synchronization mode, it is not possible to specify G27, G28, G29, G30, and G53 for a slave axis.
3. It is not possible to use controlled axis detach for a slave axis.
4. During synchronization, manual handle interruption can be performed on the slave and other axes.
5. In synchronization mode, no inch/metric conversion commands (G20 and G21) cannot be issued.
6. In synchronization mode, only the machine coordinates on a slave axis are updated.
7. If, during synchronization, G81.5 is issued again, alarm PS1594, “EGB FORMAT ERROR” is issued if bit 3 (ECN) of parameter No. 7731 is 0. If bit 3 (ECN) of parameter No. 7731 is 1, the synchronization coefficient can be changed to a newly specified one.
8. Actual cutting feedrate display does not take synchronization pulses into consideration.
9. For an EGB slave axis, synchronous and composite control cannot be executed.
10. In EGB synchronization mode, AI contour control mode is temporarily canceled.
11. Not advanced preview feed-forward but conventional feed-forward is enabled in the path where EGB synchronization mode is effective.

1.9.5.2 Description of commands compatible with those for a hobbing machine (G80, G81)

A command compatible with that for a hobbing machine can be used as a synchronization command.
Usually, a hobbing machine performs machining by synchronizing the workpiece axis (usually, the C-axis) to the hobbing axis (spindle).
If there are two synchronization sets with the EGB, which synchronization set to start with this specification method can be specified with parameter No. 7710.

Format

	Bit 0 (EFX) of parameter No.7731=0	Bit 0 (EFX) of parameter No.7731=1	
		Bit 5 (HBR) of parameter No.7731=1	Bit 5 (HBR) of parameter No.7731=0
Start of synchronization	**G81 T__ (L__) (Q__ P__) ;**	**G81.4 R__ (L__) (Q__ P__) ;**	**G81.4 T__ (L__) (Q__ P__) ;**
Cancellation of synchronization	**G80 ;**	**G80.4 ;**	**G80.4 ;**
	(*1) (*4)	**(*2) (*4)**	**(*3) (*4)**

T(or R) : Number of teeth (Specifiable range: 1 to 5000)

L : Number of hob threads (Specifiable range: -250 to 250)
The sign of L determines the direction of rotation for the workpiece axis.
When L is positive, the direction of rotation for the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation for the workpiece axis is negative (- direction).
When L is 0, it follows the setting of bit 3 (LZR) of parameter No.7701.
If L is not specified, the number of hob threads is assumed 1.

Q : Module or diametral pitch
Specify a module in the case of metric input.
(Unit: 0.00001mm, Specifiable range: 0.01 to 25.0mm)
Specify a diametral pitch in the case of inch input.
(Unit: 0.00001inch^{-1}, Specifiable range: 0.01 to 254.0 inch^{-1})

P : Gear helix angle
(Unit: 0.0001deg, Specifiable range: -90.0 to 90.0deg)

*1 Use it for machining centers.
*2 Use it for lathes.
*3 Use it for machining centers.
This format enables specification of the same G codes as those for lathes.
*4 When specifying Q and P, the user can use a decimal point.

NOTE
Specify G81, G80, G81.4, and G80.4 in a single block.

Explanation

- Synchronization start

Specify P and Q to use helical gear compensation. In this case, if only one of P and Q is specified, alarm PS1594 "EGB FORMAT ERROR" is generated.

When G81 is issued so that the machine enters synchronization mode, the synchronization of the workpiece axis to the spindle is started.

During synchronization, control is performed such that the ratio of the spindle speed to the workpiece-axis speed is the same as that of T (number of teeth) to L (number of hob threads).

If, during synchronization, G81 is issued again without synchronization cancellation, alarm PS1595 "ILL-COMMAND IN EGB MODE" is generated if ECN, bit 3 of parameter No. 7731, is 0. If ECN, bit 3 of parameter No. 7731, is 1, helical gear compensation is conducted with the synchronization coefficient being changed to the one newly specified with T and L commands if T and L commands are issued, and if T and L commands are not issued and only P and Q commands are issued, helical gear compensation is conducted with the synchronization coefficient kept intact. This allows consecutive fabrication of helical gears and spur gears.

- Synchronization cancellation

Synchronization of all synchronized axes is canceled.
When a synchronization cancellation command is issued, the absolute coordinates for the slave axis are updated according to the amount of travel during synchronization.
For a rotation axis, the value obtained by rounding off the amount of travel during synchronization to the nearest 360 degrees is added to the absolute coordinates.
In a G80 block, do not specify addresses other than O or N.

⚠ CAUTION

1 Feed hold, interlock, and machine lock are invalid to a slave axis in EGB synchronization.
2 Even if an OT alarm is issued for a slave axis in EGB synchronization, synchronization will not be canceled.
3 During synchronization, it is possible to execute a move command for a slave axis and other axes, using a program. The move command for a slave axis must be an incremental one.

NOTE

1 If bit 0 (HBR) of parameter No. 7700 is set to 1, EGB synchronization will not be canceled due to a reset. Usually, set this parameter bit to 1.
2 In synchronization mode, it is not possible to specify G27, G28, G29, G30, and G53 for a slave axis.
3 It is not possible to use controlled axis detach for a slave axis
4 During synchronization, manual handle interruption can be performed on the slave and other axes.
5 In synchronization mode, no inch/metric conversion commands (G20 and G21) cannot be issued.
6 In synchronization mode, only the machine coordinates on a slave axis are updated.
7 If bit 0 (EFX) of parameter No. 7731 is 0, no canned cycle for drilling can be used. To use a canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.
8 If bit 0 (TDP) of parameter No. 7702 is 1, the permissible range of T is 0.1 to 500 (1/10 of the specified value).
9 If, at the start of EGB synchronization (G81), L is specified as 0, synchronization starts with L assumed to be 1 if bit 3 (LZR) of parameter No.7701 is 0; if bit 3 (LZR) of parameter No.7701 is 1, synchronization is not started with L assumed to be 0. At this time, helical gear compensation is performed.
10 Feed per revolution is performed on the feedback pulses on the spindle. By setting bit 0 (ERV) of parameter No. 7703 to 1, feed per revolution can be performed based on the speed on the synchronous slave axis.
11 Actual cutting feedrate display does not take synchronization pulses into consideration.
12 For an EGB slave axis, synchronous and composite control cannot be executed.
13 In EGB synchronization mode, AI contour control mode is temporarily canceled.
14 Not advanced preview feed-forward but conventional feed-forward is enabled in the path where EGB synchronization mode is effective.

- Helical gear compensation

For a helical gear, the workpiece axis is subjected to compensation for movement along the Z axis (axial feed axis) according to the twisted angle of the gear.
Helical gear compensation is performed with the following data.

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (for metric input)}$$

$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (for inch input)}$$

where

Compensation angle : Absolute value with sign (degrees)
Z : Amount of travel along the Z axis after a G81 command is issued (mm or inch)
P : Twisted angle of the gear with sign (degrees)
π : Circular constant
T : Number of teeth
Q : Module (mm) or diametral pitch (inch^{-1})

Use P, T, and Q specified in the G81 block.
In helical gear compensation, the machine coordinates on the workpiece axis and the absolute coordinates are updated with helical gear compensation.
The updated timing is as follows.

<table>
<tr><th>Machine coordinates</th><th colspan="2">Absolute coordinates</th></tr>
<tr><td rowspan="3">During helical gear compensation</td><td colspan="2">Bit 7(HAD) of parameter No.7731</td></tr>
<tr><td>= 0</td><td>= 1</td></tr>
<tr><td>Synchronization cancel</td><td>During helical gear compensation</td></tr>
</table>

- Direction of helical gear compensation

The direction depends on bit 2 (HDR) of parameter No. 7700.

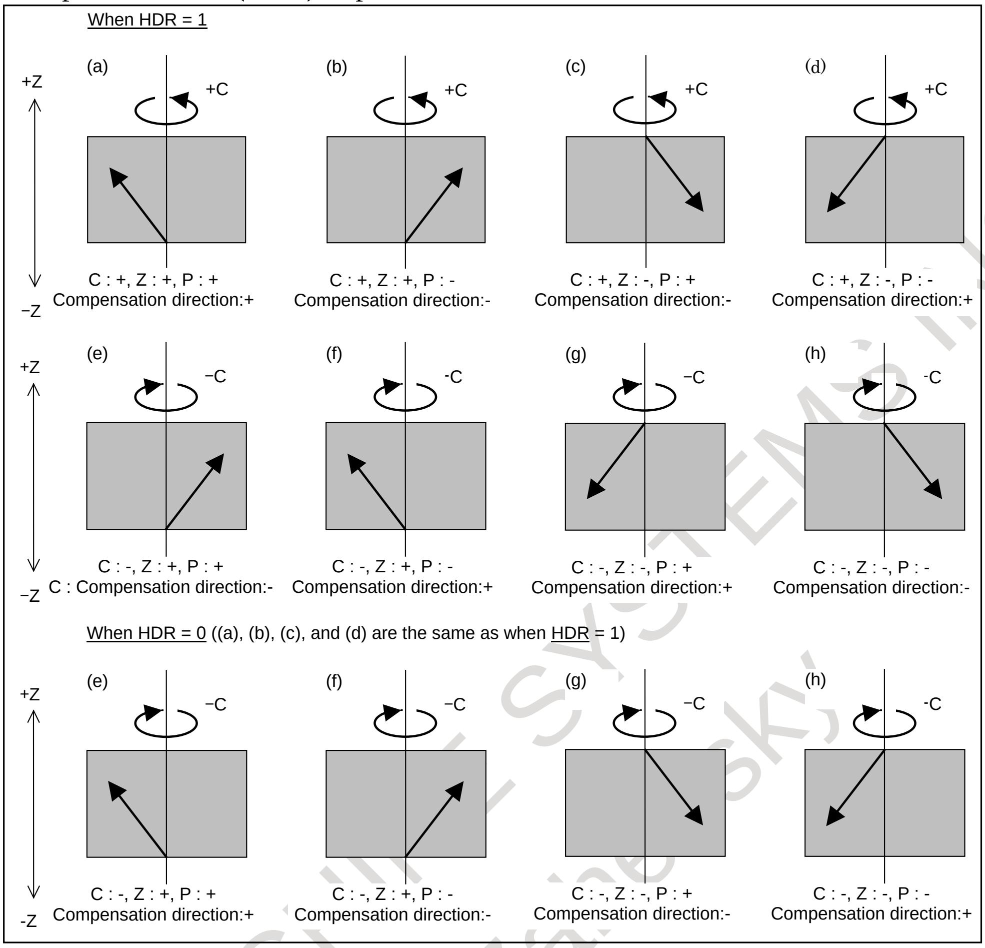


Fig. 1.9.5.2 (a) Direction of helical gear compensation

1.9.5.3 Controlled axis configuration example

- For gear grinders

Spindle : EGB master axis : Tool axis
1st axis : X axis
2nd axis : Y axis
3rd axis : C axis (EGB slave axis : Workpiece axis)
4th axis : C axis (EGB dummy axis : Cannot be used as a normal controlled axis)
5th axis : V axis (EGB slave axis : Dressing axis)
6th axis : V axis (EGB dummy axis : Cannot be used as a normal controlled axis)

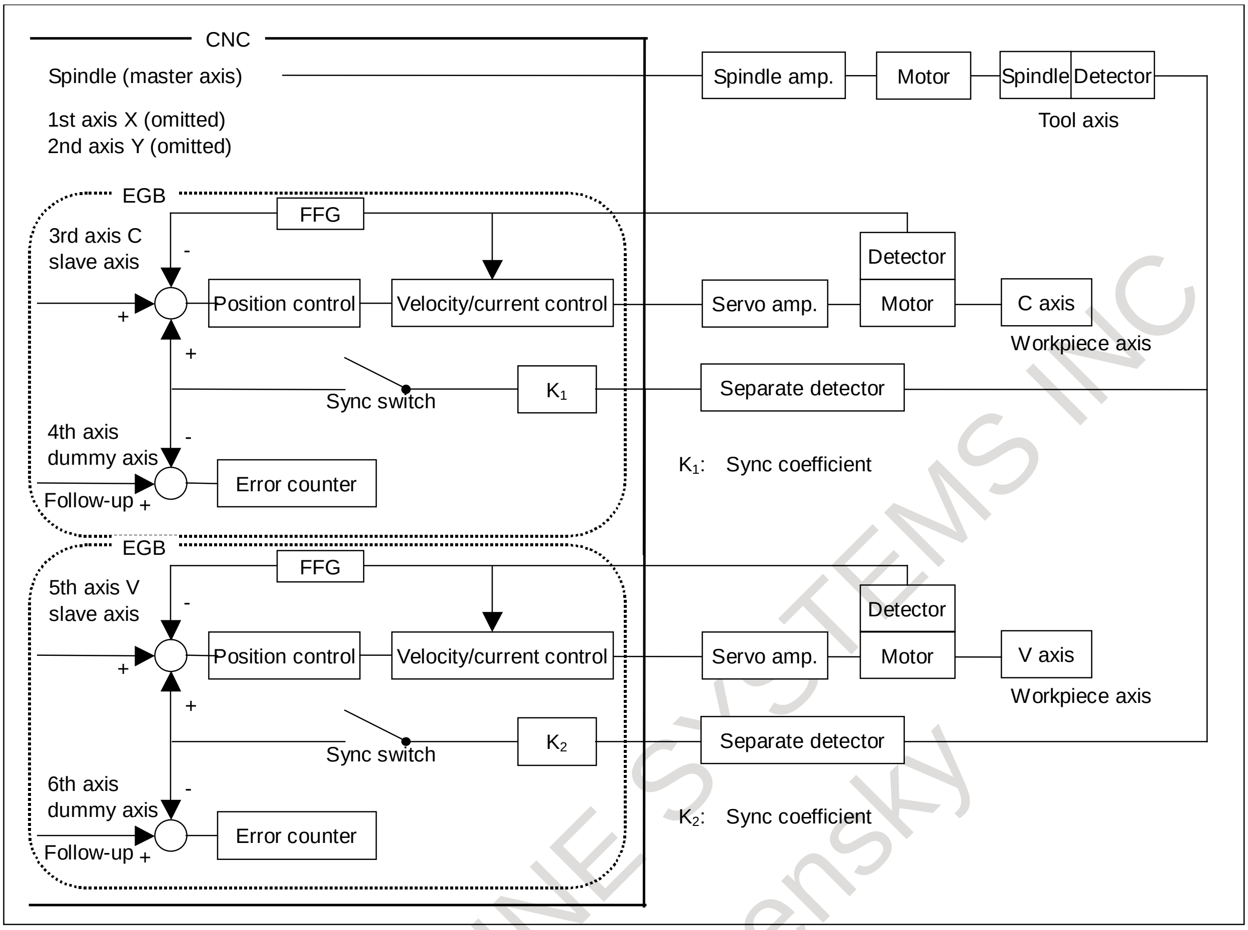


Fig. 1.9.5.3 (a)

For EGB axis configuration parameter setting examples, see the section on "FSSB setting".

- Example of use of dressing

Gear grinder in the following machine configuration

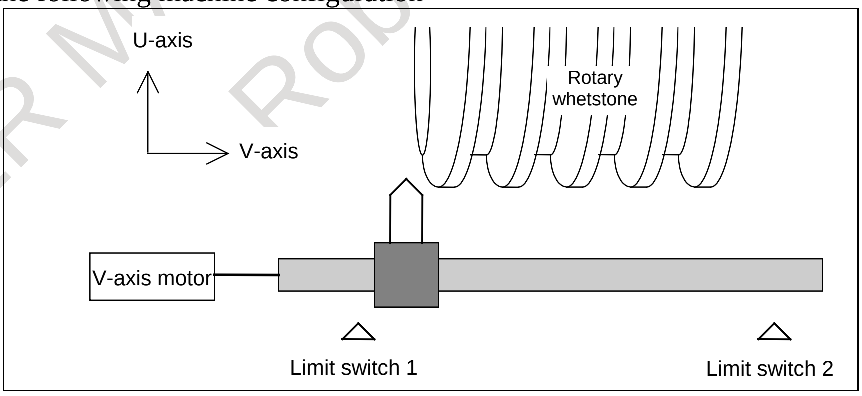


Fig. 1.9.5.3 (b)

O9500 ;

N01 G01 G91 U_ F100 ;	Dressing axis approach
N02 M03 S100 ;	The M03 command causes the PMC to rotate the whetstone in the positive direction. In accordance with this, the tool moves along the V-axis in the + direction. When the tool reaches the position of limit switch 2 on the V-axis, the PMC stops the whetstone and returns FIN.
N02 U_ V_ ;	Movement to the next dressing position

N03 M04 S100 ; The M04 command causes the PMC to rotate the whetstone in the negative direction.
In accordance with this, the tool moves along the V-axis in the - direction. When the tool reaches the position of limit switch 1 on the V-axis, the PMC stops the whetstone and returns FIN.

N04 U_ V_ ; Movement to the next dressing position
If required, N02 to N04 are repeated to conduct dressing.

....
....
M99 ;

> **NOTE**
> If the V-axis (linear axis) is synchronized with the spindle as in dressing, the V-axis travel range is determined by the rotation of the spindle. To perform dressing with the tool moving back and forth along the V-axis in a certain range, therefore, the PMC must perform an operation in which the tool is stopped temporarily and is reversed when it reaches a certain position on the V-axis.
> In the above example, limit switches are provided to determine the range of travel along the V-axis and the PMC performs control so that the whetstone rotates until the tool reaches the position of each limit switch on the V-axis.
> By using the position switch function instead of limit switches, dressing can be performed as in the following example, without the need to mount limit switches to the machine. By rewriting the operating ranges of the position switches (parameters Nos. 6930 to 6945 and 6950 to 6965) using the G10 programmable parameter input, the range of travel along the V-axis can be specified using a program.

1.9.5.4 Retract function

See Item, "Retract function" in the Subsection "Electronic Gear Box".

Signal

Retract signal RTRCT<Gn066.4>

[Classification] Input signal
[Function] Retracts along the axis specified in the parameter.
[Operation] When this signal is set to “1”, the CNC operates in the following way.
At the rising edge of this signal, retraction can be performed for the axis for which a retract value is set in parameter No. 7741. The retract value and retract feedrate set in parameters Nos. 7741 and 7740 are used. Upon the completion of retraction, retract completion signal RTRCTF <Fn065.4> is output. The retract signal is valid in either automatic operation mode (MEM, MDI, etc.) or manual operation mode (HND, JOG, etc.). When the retract signal is set to “1” during automatic operation, retraction is performed and automatic operation is stopped.

Retraction completion signal RTRCTF<Fn065.4>

[Classification] Output signal
[Function] Reports that retraction is finished.
[Operation] This signal is set to “1” in the following case.
- When retraction is finished (movement is finished)

This signal is set to “0” in the following case.
- When a move command is issued for any retract axis after the end of a retract operation.

> **NOTE**
> When the retraction completion signal is "1," the retract signal is not accepted.

EGB mode signal SYNMOD< Fn065.6>

[Classification] Output signal

[Function] Reports that synchronization with the EGB is in progress.

[Operation] This signal becomes “1” if:

- Synchronization with the EGB is in progress.

It becomes “0” if:

- Synchronization with the EGB is canceled.

EGB mode confirmation signal EGBM1<Fn208.0>, EGBM2<Fn208.1>, …

[Classification] Output signal

[Function] Reports that synchronization is being executed by EGB. This signal is output to a slave axis.

EGBMx

x : 1 ..... First axis synchronized by EGB
2 ..... Second axis synchronized by EGB
3 ..... Third axis synchronized by EGB
: :
: :

[Operation] This signal is set to “1” in the following case.

- During synchronization caused by EGB

This signal is set to “0” in the following case.

- When synchronization caused by EGB is released

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn066				RTRCT				
Fn065		SYNMOD		RTRCTF				
Fn208	EGBM8	EGBM7	EGBM6	EGBM5	EGBM4	EGBM3	EGBM2	EGBM1

Parameter

The following table lists the parameters related to EGB.

Parameter number	Description
1006#0 1006#1	To specify the speed for the slave axis by L in synchronization command, an EGB slave axis and an EGB dummy axis require that the setting of a rotary axis (type A) (bit 0 (ROT) of parameter No. 1006 be 1 and bit 1 (ROS) of parameter No. 1006 be 0).
1023	Set from the FSSB setting screen. For FSSB manual setting, be sure to set the EGB axis as described below: The slave axis must be set with an odd number, and the dummy axis with an even number. They must be consecutive. Example: If the servo axis number of the slave axis is 1, the servo axis number of the dummy axis must be set to 2. If the servo axis number of the slave axis is 3, the servo axis number of the dummy axis must be set to 4.
2011 #0	Specify an axis to be synchronized. Specify 1 for both an EGB slave axis and EGB dummy axis.
3115#0	The current position is not indicated for an axis for which this parameter is set to 1.Since the current position for an EGB dummy axis has no meaning, set this parameter to 1 to delete the current position indication for the axis from the screen.
7700#0	The synchronization mode is canceled (0)/not canceled by a reset (1).

Parameter number	Description
7700#2	Compensation direction for helical gear compensation
7701#3	At the start of synchronization (G81), synchronization is started (0)/not started (1) if the number of hob threads L is specified as 0.
7702#0	The specifiable number of teeth, T, at the start of synchronization (G81) is not reduced to a 1/10 of a specified value (0)/reduced (1).
7702#3	The retract function with an alarm is disabled (0)/enabled (1).
7703#0	During synchronization (G81), feed per revolution is performed for feedback pulses (0)/pulses converted to the speed for the workpiece axis(1).
7703#1,#2	Specify when to perform a retract operation with the retract function with an alarm; during synchronization; during synchronization and automatic operation; or during synchronization or automatic operation.
7709	Number of the axial feed axis in helical gear compensation
7710	Axis number of an axis to be synchronized using the method of command specification for a hobbing machine
7731#0	The EGB command is G80 and G81(0)/G80.4 and G81.4(1).
7731#3	When the automatic phase synchronization function for the electronic gear box is disabled, the G81 command cannot be issued again (an alarm is issued) (0)/can be issued again (1)during EGB synchronization.
7731#5	In EGB synchronization start command G81.4, the number of teeth is specified in T (0)/specified in R (1).
7740	Feedrate during retraction
7741	Retract amount
7772	Number of position detector pulses per rotation about tool axis
7773	Number of position detector pulses per rotation about workpiece axis
7782	Pulse count of position detector per rotation about EGB master axis
7783	Pulse count of position detector per rotation about EGB slave axis

For FSSB settings, see the section on “FSSB settings”.
If FSSB setting mode is automatic setting mode, setting is made automatically by inputting data to the FSSB setting screen. For the slave/dummy axes of EGB, set the value in the “M/S” item in the FSSB axis setting screen same way of the tandem setting.
Note the following points when specifying parameters for the electronic gear box.

1 Specify an axis that is not used or the same name as that for a slave axis for the name of a dummy axis. Do not use a name which is usually not allowed to be used as an axis address, such as D.
2 Specify the same values for an EGB slave axis and an EGB dummy axis in the following parameters.
   1013#0 to 3 Increment system
   1004#7 Ten times minimum input increment
   1006#0,1 Rotary axis setting
   1006#3 Diameter/radius specification
   1420 Rapid traverse rate
   1421 Rapid-traverse override F0 speed
   1820 Command multiplication
   2000 and over Parameters related to digital servo
3 To specify the speed for the slave axis by L in the synchronization command, set parameter No. 1260 (amount of travel per rotation about a rotation axis) for the slave and dummy axes.
4 Make the specification for a dummy axis in the following way.
   1815#1 Whether to use separate detectors. Although an EGB dummy axis uses the interface of a separate detector, set these parameters to 0.
5 Reducing synchronous errors requires enabling the feed-forward function for the slave axis.
   For details, see “How to reduce the synchronous error” in “Explanation” of this chapter.
6 Reducing shocks that may occur at the beginning of acceleration/deceleration requires enabling the soft start/stop function for the spindle axis. For details, see “How to reduce shock at the start of acc./dec.” in “Explanation” of this chapter.

1023	Number of the servo axis for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 80

This parameter associates each control axis with a specific servo axis. Specify values 1+8n, 2+8n, 3+8n, 4+8n, 5+8n, and 6+8n (n = 0, 1, 2, …, 9) like 1, 2, 3, 4, 5, …, 77, and 78.
The control axis number is the order number that is used for setting the axis-type parameters or axis-type machine signals
For electronic gear box (EGB) controlled axes, two axes need to be specified as one pair. So, make a setting as described below.
For a slave axis, set an odd (1, 3, 5, 7, 9, ...) servo axis number. For a dummy axis to be paired, set a value obtained by adding 1 to the value set for the slave axis.

	#7	#6	#5	#4	#3	#2	#1	#0
2011								SYNx

[Input type] Parameter input
[Data type] Bit axis

#0 SYNx When the electronic gear box function (EGB) is used, this bit sets the axis to be synchronized.
0: Axis not synchronized by EGB
1: Axis synchronized by EGB
Set 1 for both of the slave and dummy axes of EGB.

> **NOTE**
> The setting of this parameter becomes valid after the power is turned off then back on.

	#7	#6	#5	#4	#3	#2	#1	#0
3115								NDPx

[Input type] Parameter input
[Data type] Bit axis

#0 NDPx The current position is:
0: Displayed.
1: Not displayed.

> **NOTE**
> When using the electronic gear box (EGB) function, set 1 for the EGB dummy axis to disable current position display.

	#7	#6	#5	#4	#3	#2	#1	#0
7700						**HDR**		**HBR**

[Input type] Parameter input
[Data type] Bit path

#0 HBR When the electronic gear box (EGB) function is used, performing a reset:
0: Cancels the synchronization mode (G81 or G81.5).
1: Does not cancel the synchronization mode. The mode is canceled only by the G80 or G80.5 command.

#2 HDR Direction of helical gear compensation (usually, set 1.)
(Example) To cut a left-twisted helical gear when the direction of rotation about the C-axis is the negative (-) direction:
0: Set a negative (-) value in P.
1: Set a positive (+) value in P.

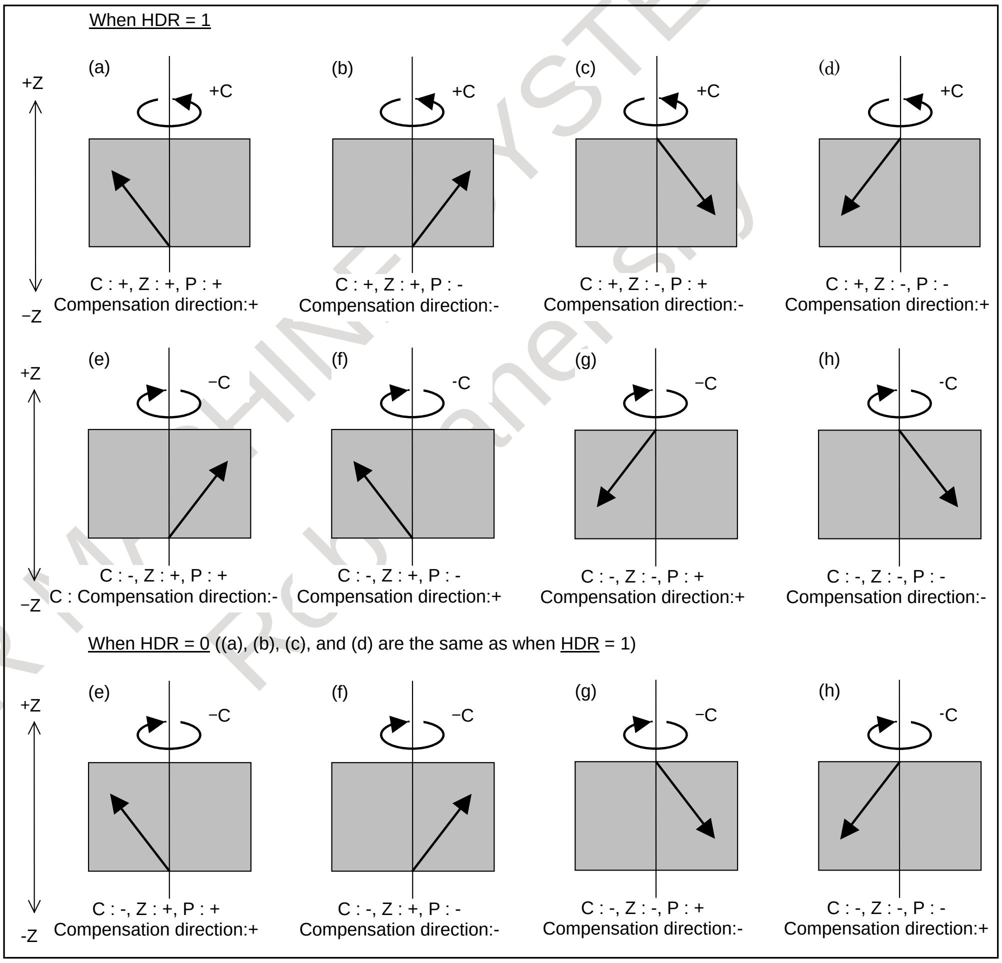


Fig. 1.9.5.4 (a) Direction of helical gear compensation

	#7	#6	#5	#4	#3	#2	#1	#0
7701					**LZR**			

[Input type] Parameter input
[Data type] Bit path

#3 LZR When L (number of hob threads) = 0 is specified at the start of EGB synchronization (G81):

0: Synchronization is started, assuming that L = 1 is specified.
1: Synchronization is not started, assuming that L = 0 is specified. However, helical gear compensation is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
7702					ART			TDP

[Input type] Parameter input
[Data type] Bit path

#0 TDP The specifiable number of teeth, T, of the electronic gear box (G81) is:
0: 1 to 5000
1: 0.1 to 500 (1/10 of a specified value)

> **NOTE**
> In either case, a value from 1 to 5000 can be specified.

#3 ART The retract function executed when an alarm is issued is:
0: Disabled.
1: Enabled.
When an alarm is issued, a retract operation is performed with a set feedrate and travel distance (parameters Nos. 7740 and 7741).

> **NOTE**
> If a servo alarm is issued for other than the axis along which a retract operation is performed, the servo activating current is maintained until the retract operation is completed.

	#7	#6	#5	#4	#3	#2	#1	#0
7703						ARO	ARE	ERV

[Input type] Parameter input
[Data type] Bit path

#0 ERV During EGB synchronization (G81), feed per revolution is performed for:
0: Feedback pulses.
1: Pulses converted to the speed for the workpiece axis.

#1 ARE The retract function executed when an alarm is issued retracts the tool during:
0: EGB synchronization or automatic operation (automatic operation signal OP <Fn000.7> = “1”).
1: EGB synchronization.

#2 ARO The retract function executed when an alarm is issued retracts the tool during:
0: EGB synchronization.
1: EGB synchronization and automatic operation (automatic operation signal OP <Fn000.7> = “1”).

> **NOTE**
> This parameter is valid when bit 1 (ARE) of parameter No. 7703 is set to 1.

The following table lists the parameter settings and corresponding operation.

ARE	ARO	Operation
1	0	During EGB synchronization
1	1	During EGB synchronization and automatic operation
0	0	During EGB synchronization or automatic operation
0	1	

> **NOTE**
> Parameters ARE and ARO are valid when bit 3 (ART) of parameter No. 7702 is set to 1 (when the retract function executed when an alarm is issued).

7709	Number of the axial feed axis for helical gear compensation

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to Number of controlled axes

This parameter sets the number of the axial feed axis for a helical gear.

> **NOTE**
> When this parameter is set to 0 or a value outside the valid setting range, the Z-axis becomes the axial feed axis.

7710	Axis number of an axis to be synchronized using the method of command specification for a hobbing machine

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to Number of controlled axes

When there are several groups of axes to be synchronized (the axes for which bit 0 (SYNMOD) of parameter No. 2011 is set to 1), an axis for which to start synchronization is specified using the following command (for a hobbing machine):

G81 T t L ± *l* ;

t: Spindle speed ($1 \le t \le 5000$)

l: Number of synchronized axis rotations ($-250 \le l \le 250$)

Synchronization between the spindle and a specified axis is established with the ratio of $\pm l$ rotations about the synchronized axis to t spindle rotations.

t and *l* correspond to the number of teeth and the number of threads on the hobbing machine, respectively.

Above command is issued without setting this parameter when there are several groups of axes to be synchronized, the alarm PS1593, “EGB PARAMETER SETTING ERROR” is issued.

> **NOTE**
> 1 Set this parameter when there are two or more groups of servo and spindle EGBs in the same path. Set 0 when there is one group of EGBs in the same path.
> 2 When there are two or more groups of servo and spindle EGBs in the same path, setting a value outside the valid data range in this parameter causes alarm PS1593 to be issued.
> 3 For Series 16*i*, when a value outside the valid data range is set in this parameter, the fourth axis is assumed according to the specifications.

NOTE
4 The setting of this parameter becomes valid after the power is turned off then back on.

	#7	#6	#5	#4	#3	#2	#1	#0
7731	HAD		HBR		ECN			EFX

[Input type] Parameter input
[Data type] Bit path

#0 EFX As the EGB command:
0: G80 and G81 are used.
1: G80.4 and G81.4 are used.

NOTE
When this parameter is set to 0, no canned cycle for drilling can be used.

#3 ECN When the automatic phase synchronization function for the electronic gear box is disabled, during EGB synchronization, the G81 or G81.5 command:
0: Cannot be issued again. (The alarm PS1595, "ILL-COMMAND IN EGB MODE" is issued.)
1: Can be issued again.

#5 HBR In EGB synchronization start command G81.4, the number of teeth is:
0: Specified in T.
1: Specified in R.

NOTE
This parameter is valid when bit 0 (EFX) of parameter No. 7731 is set to 1.

#7 HAD In electronic gear box, the timing for reflecting helical gear compensation and travel distance of automatic phase synchronization to absolute coordinates is:
0: When synchronization is canceled.
1: During helical gear compensation and automatic phase synchronization.

7740	Feedrate during retraction

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the feedrate during retraction for each axis.

7741	Retract amount

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the retract amount for each axis.

7772	Number of position detector pulses per rotation about the tool axis

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[Valid data range] 1 to 999999999

This parameter sets the number of pulses per rotation about the tool axis (on the spindle side), for the position detector.

For an A/B phase detector, set this parameter with four pulses equaling one A/B phase cycle.

7773	Number of position detector pulses per rotation about the workpiece axis

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[Valid data range] 1 to 999999999

This parameter sets the number of pulses per rotation about the workpiece axis (on the slave side), for the position detector.

Set the number of pulses output by the detection unit.

Set parameters Nos. 7772 and 7773 when using the G81 EGB synchronization command.

[Example 1] When the EGB master axis is the spindle and the EGB slave axis is the C-axis

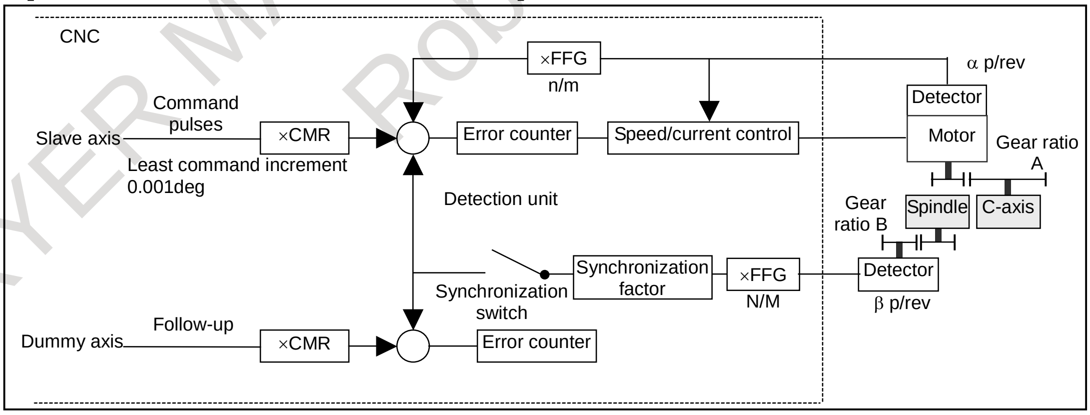


Fig. 1.9.5.4 (b)

Gear ratio of the spindle to the detector B:
    1/1 (The spindle and detector are directly connected to each other.)
Number of detector pulses per spindle rotation β: 80,000 pulses/rev
(Calculated for four pulses for one A/B phase cycle)
FFG N/M of the EGB dummy axis: 1/1
Gear ratio of the C-axis A: 1/36 (One rotation about the C-axis to 36 motor rotations)
Number of detector pulses per C-axis rotation α: 1,000,000 pulses/rev

C-axis CMR: 1
C-axis FFG n/m: 1/100

In this case, the number of pulses per spindle rotation is:
$80000 \times 1/1 = 80000$
Therefore, set 80000 for parameter No. 7772.
The number of pulses per C-axis rotation in the detection unit is:

$$1000000 \div \frac{1}{36} \times \frac{1}{100} = 360000$$

Therefore, set 360000 for parameter No. 7773.

[Example 2] When the gear ratio of the spindle to the detector B is 2/3 for the above example (When the detector rotates twice for three spindle rotations)
In this case, the number of pulses per spindle rotation is:

$$80000 \times \frac{2}{3} = \frac{160000}{3}$$

160000 cannot be divided by 3 without a remainder. In this case, change the setting of parameter No. 7773 so that the ratio of the settings of parameters Nos. 7772 and 7773 indicates the value you want to set.

$$\frac{\text{No.7772}}{\text{No.7773}} = \frac{160000/3}{360000} = \frac{160000}{360000 \times 3} = \frac{160000}{1080000}$$

Therefore, set 160000 for parameter No. 7772 and 1080000 for parameter No. 7773.
As described above, all the settings of parameters Nos. 7772 and 7773 have to do is to indicate the ratio correctly. So, you can reduce the fraction indicated by the settings. For example, you may set 16 for parameter No. 7772 and 108 for parameter No. 7773 for this case.

7782	Number of pulses from the position detector per EGB master axis rotation

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 999999999

For a slave axis, set the number of pulses generated from the position detector per EGB master axis rotation.
For an A/B phase detector, set this parameter with four pulses equaling one A/B phase cycle.

7783	Number of pulses from the position detector per EGB slave axis rotation

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 999999999

For a slave axis, set the number of pulses generated from the position detector per EGB slave axis rotation.
Set the number of pulses output by the detection unit.
Set this parameter when using the G81.5 EGB synchronization command.
The method for setting parameters Nos. 7782 and 7783 is the same as for parameters Nos. 7772 and 7773. For the method, see the description of parameters Nos. 7772 and 7773.

The ratio of the number of pulses for the master slave to that of pulses for the slave axis may be valid, but the settings of the parameters may not indicate the actual number of pulses. For example, the number of pulses may not be able to be divided without a remainder for the reason of the master and slave axis gear ratios as described in example 2. In this case, the following methods cannot be used for the G81.5 command:

G81.5 T_ C_ ; When the speed is specified for the master axis and the travel distance is specified for the slave axis
G81.5 P_ C0 L_ ; When the number of pulses is specified for the master axis and the speed is specified for the slave axis

	#7	#6	#5	#4	#3	#2	#1	#0
2005							FEEDx	

[Input type] Parameter input
[Data type] Bit axis

#1 FEEDx Feed-forward function is:
0: Invalid.
1: Valid.

Set 1 for the EGB slave axis.

> **NOTE**
> This parameter is Initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

2068	Feed-forward function coefficient

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 100
Setting value = $\alpha \times 100$

Set 10000 for the EGB slave axis.

	#7	#6	#5	#4	#3	#2	#1	#0
2273		EGFx						

[Input type] Parameter input
[Data type] Bit axis

#6 EGFx FFG is:
0: Not considered in the synchronization coefficient.
1: Considered.

The synchronization coefficient is subject to the following restriction:

Synchronization coefficient $= \frac{L}{T} \times \frac{\beta}{\alpha}$

where $\frac{\mathrm{L}}{\mathrm{T}} \times \frac{\beta}{\alpha} \leq \frac{\mathrm{2word}}{\mathrm{1word}}$: Condition <1>

where
L : Number of hob threads
T : Number of teeth

α : Number of pulses of the position detector per rotation about the master axis (parameter No. 7772 or 7782)
β : Number of pulses of the position detector per rotation about the slave axis (parameter No. 7773 or 7783)

If this condition, <1>, cannot be satisfied, set this parameter bit to 1. With this setting, FFG is considered in the synchronization coefficient, and by selecting FFG appropriately, it is possible to set α and β in such a way that condition <1> can be satisfied with the synchronization coefficient kept intact.

Synchronization coefficient $= \frac{L}{T} \times \frac{\beta}{\alpha} \times \frac{N}{M}$

where $\frac{L}{T} \times \frac{\beta}{\alpha} \leq \frac{2word}{1word}$: Condition <1>

N: Numerator of FFG
M: Denominator of FFG

The new value of α is the old one multiplied by FFG.

$$\alpha[\text{New}] = \alpha[\text{Old}] \times \frac{N}{M}$$

Setting example

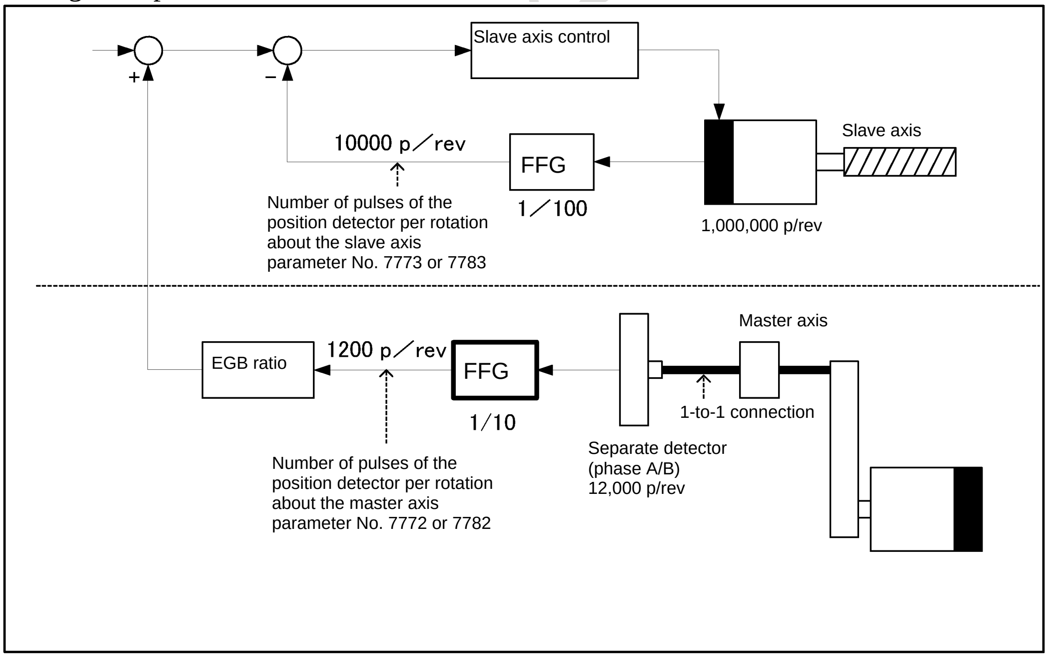


Fig. 1.9.5.4 (c)

Master axis conditions:
The separate detector must be 12000 p/rev.
The master axis and the separate detector must have a 1-to-1 connection.

Slave axis conditions:
The motor Pulsecoder must be 1 million p/rev.
FFG must be 1/100.

Determine FFG so that condition <1> is satisfied.

$\frac{L}{T} \times \frac{\beta}{\alpha} \leq \frac{2\text{word}}{1\text{word}}$: Condition <1>

In this example, FFG is set to 1/10. Set bit 6 (EGF) of parameter No. 2273, which is a function bit to consider FFG in EGB, to 1, and set the number of pulses of the position detector per rotation about the master and slave axes.

As the number of pulses of the position detector per rotation about the master axis, set 12000 x FFG (1/10) = 1200.
As the number of pulses of the position detector per rotation about the slave axis, set 10000.

	#7	#6	#5	#4	#3	#2	#1	#0
4399						**SOSALWs**		

[Input type] Parameter input
[Data type] Bit spindle

#2 SOSALWs The Soft start/stop function is:
0: Disabled at an emergency stop (*ESP is ”0”) or MRDY is ”0”.
1: Enabled even at an emergency stop (*ESP is ”0”) or MRDY is ”0”.

4030	**Soft start/stop setting time**

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] $1min^{-1}/sec$
[Valid data range] 0 to 32767

This parameter sets an acceleration (rate at which the speed changes) applied when the Soft start/stop function is enabled (the Soft start/stop signal SOCNA <Gn071> is ”1”).

4508	**Acceleration rate change applied at Soft start/stop**

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] $10min^{-1}/sec^2$
[Valid data range] 0 to 32767

This parameter sets an acceleration rate (rate at which the acceleration changes) applied when the Soft start/stop function is enabled (the Soft start/stop signal SOCNA <Gn071> is ”1”).

> **NOTE**
> If the setting of this parameter is 0, it means that the speed to be applied when the Soft start/stop function is enabled is linear.

Alarm and message

Number	Message	Description
PS1593	EGB PARAMETER SETTING ERROR	Error in setting a parameter related to the EGB (1) The setting of bit 0 (SYNx) of parameter No. 2011, is not correct. (2) The slave axis specified with G81 is not set as a rotation axis. (bit 0 (ROTx) of parameter No. 1006) (3) Number of pulses per rotation (Parameter (No. 7772 or 7773) or (No. 7782 or 7783)) is not set. (4) Parameter No. 7710 is not set for a command compatible with that for a hobbing machine.

Number	Message	Description
PS1594	EGB FORMAT ERROR	Error in the format of the block of an EGB command (1) T (number of teeth) is not specified in the G81 block. (2) In the G81 block, the data specified for one of T, L, P, and Q is out of its valid range. (3) In the G81 block, only one of P and Q is specified. (4) In the G81.5 block, no command is specified for the master or slave axis. (5) In the G81.5 block, data outside the valid data range is specified for the master or slave axis.
PS1595	ILL-COMMAND IN EGB MODE	During synchronization with the EGB, a command that must not be issued is issued. (1) Slave axis command using G27, G28, G29, G30, G33, G53, etc. (2) Inch/metric conversion command using G20, G21, etc. (3) Synchronization start command using G81 or G81.5 when bit 3 (ECN) of parameter No. 7731 is 0
PS1596	EGB OVERFLOW	An overflow occurred in the calculation of the synchronization coefficient.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Electronic gear box 2 pair

1.9.6 U-axis Control

Overview

To control an axis on a spindle such as the U-axis of a vertical lathe from a motor mounted on other than the spindle, a mechanism was conventionally required which used a planetary gear box, differential gears, and others to prevent the movement along the U-axis as the spindle rotated.
The U-axis control function enables the tool to remain in a fixed position on the U-axis or to move at a programmed feedrate without using a mechanism such as a planetary gear box. This is done by causing the U-axis motor to rotate in such a way that the movement along the U-axis, which should be caused by rotation of the spindle, is canceled out.

An electronic gear box (EGB) is used to cause the U-axis motor to rotate in synchronization with the spindle.
With the EGB, the servo CPU processes signals received from the position coder mounted on the spindle at high speed to control the movement along the U-axis. It is capable of high-precision synchronous control. The EGB requires axis control circuits for two axes (U-axis and U’-axis). It acquires the pulses necessary for synchronization from the separate feedback connector on the U’-axis side. With the EGB, the spindle, which is used as the reference for synchronization, is called the master axis. The U-axis, along which the tool moves in synchronization with the master axis, is called the slave axis. The U’-axis, which acquires the pulses necessary for synchronization, is called the dummy axis.

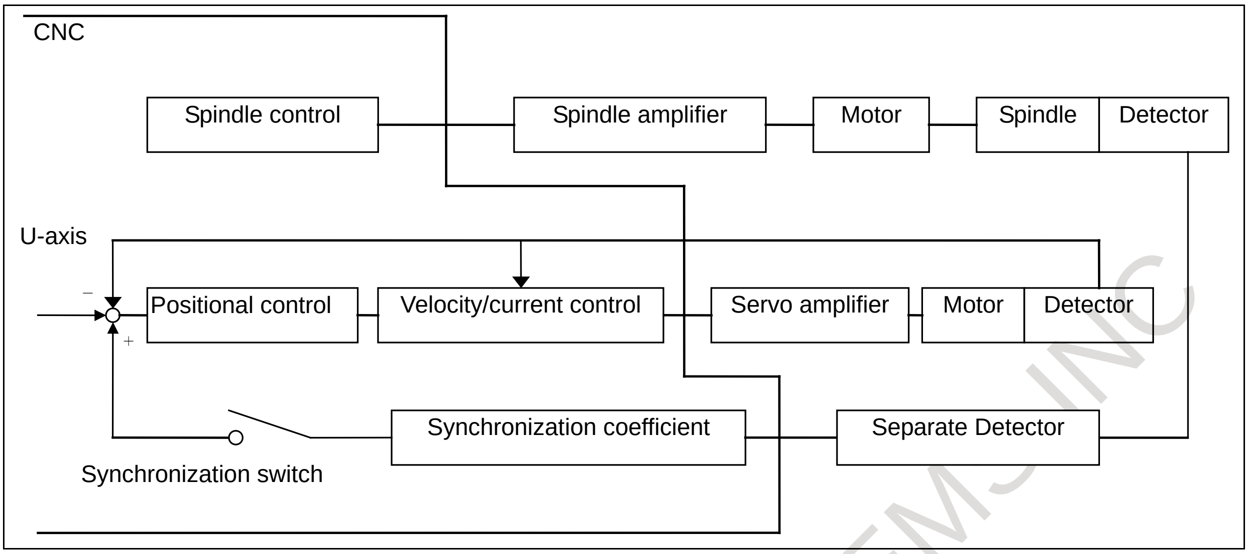


Fig. 1.9.6 (a) Block diagram of U-axis control

Example

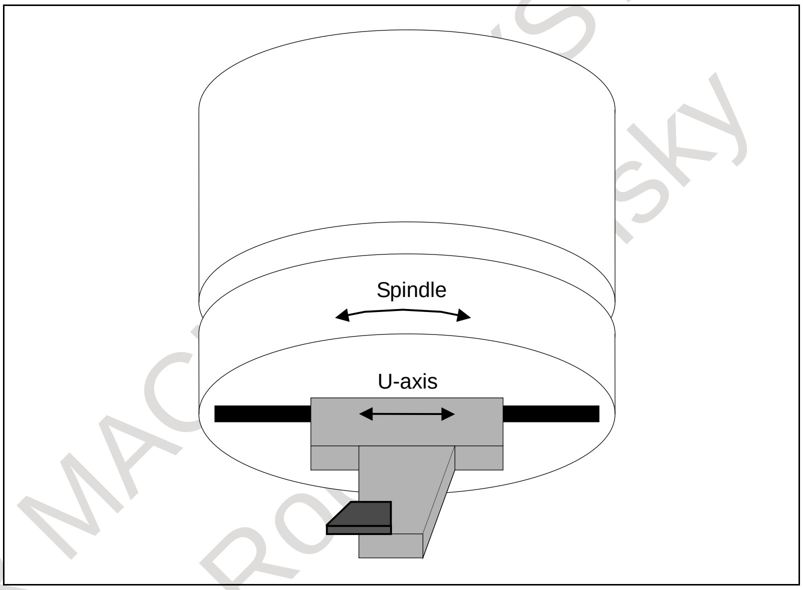


Fig. 1.9.6 (b) Example of a machine having the U-axis

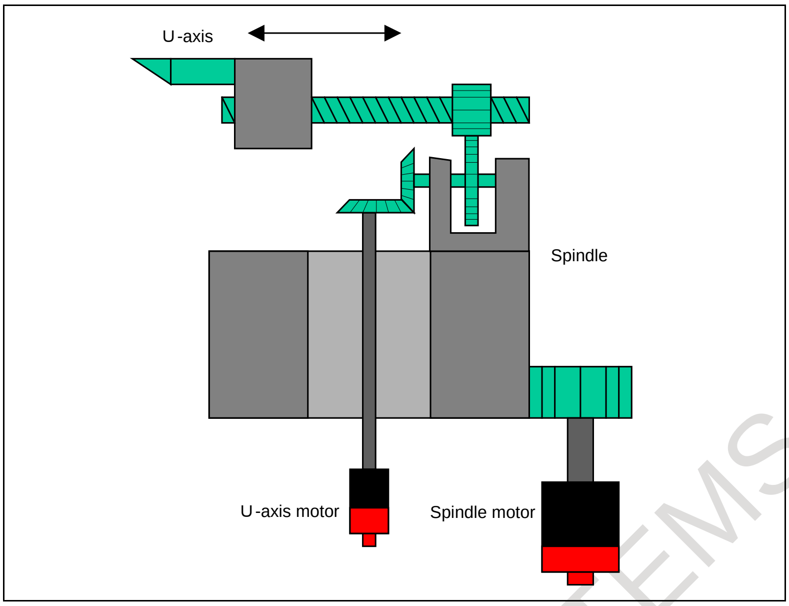


Fig. 1.9.6 (c) Example of the structure of a machine having the U-axis

In the example of the above structure, the tool moves along the U-axis when the spindle rotates. This movement is canceled out by rotating the U-axis motor.

Explanation

- Synchronization start

Synchronization is started when the EGB synchronization mode selection signal EGBS<Gn067.4> is set to "1". In the synchronization mode, the EGB synchronization mode confirmation signal EGBSM<Fn082.6> becomes "1".

If the U-axis motor rotates in synchronization with the spindle in the synchronization mode, the machine coordinates are not updated, that is, the machine coordinates indicate the position on the U-axis based on the spindle.

- Synchronization cancellation

Synchronization is canceled when the EGB synchronization mode selection signal EGBS<Gn067.4> is set to "0". When synchronization is canceled, the EGB synchronization mode confirmation signal EGBSM<Fn082.6> also becomes "0".

- Parameter Setting

(1) Set the following parameters:
- U-axis control enable: Bit 1 (UAX) of parameter No. 7702
- Number of pulses per rotation about the spindle: Parameter No. 7772
- Amount of travel by the U-axis motor per rotation about the spindle: Parameter No. 7773
- EGB function enable: Bit 0 (SYNx) of parameter No. 2011
- Parameters for feed forward control

(2) Be sure to set bit 0 (SYNx) of parameter No. 2011 to 1 for both the U-axis and dummy axis.

(3) Set the servo parameters for the dummy axis (parameters Nos. 2000 to 2999) such that they are consistent with the settings made for the U-axis.

(4) Be sure to set the command multiplier (CMR) for the dummy axis to the same value as for the U-axis.

(5) The following parameters need not be set for the dummy axis:
- Reference counter size: Parameter No. 1821
- In-position width: Parameter No. 1826
- Positioning deviation limit in movement/in the stopped state: Parameters Nos. 1828 and 1829
- Stored stroke limits: Parameter No. 1320 and others

(6) Set the values related to the flexible feed gear ratio for the dummy axis (parameters Nos. 2084 and 2085) to 1.
(7) The dummy axis occupies one servo axis interface. Set servo axis numbers consecutively for the dummy axis and U-axis so that an odd number and an even number are assigned.

Example 1)
When the dummy axis is the 4th out of total four axes:
1st axis Parameter No. 1023: 1
2nd axis Parameter No. 1023: 2
3rd axis Parameter No. 1023: 3 (U-axis)
4th axis Parameter No. 1023: 4 (dummy axis)

Example 2)
When the dummy axis is the 5th out of total five axes:
1st axis Parameter No. 1023: 1
2nd axis Parameter No. 1023: 2
3rd axis Parameter No. 1023: 5
4th axis Parameter No. 1023: 3 (U-axis)
5th axis Parameter No. 1023: 4 (dummy axis)

(8) Set bit 0 (HBR) of parameter No. 7700 to 1 so as not to cancel synchronization by the reset operation.
(9) Set bit 1 (UFF) of parameter No. 7786 to 1, when a interpolation command to between the U-axis and the other axes is specified while U-axis synchronization.
(10) Parameter setting related to feed forward control

Step 1.
Modify the motor type for the U-axis and dummy axis and set them automatically.
- Parameter No. 2020 = Motor number
- Bit 1 (DGPx) of parameter No. 2000 = 0

Set the above parameters, then turn the power off, then on again.

Step 2.
Set the parameters related to the EGB again.
- Bit 0 (SYNx) of parameter No. 2011 = 1 (for both the U-axis and dummy axis)
  Bit 1 (FFAL) of parameter No. 2011 = 1, when feed forward control is also enabled in rapid traverse.

Step 3.
Other parameters (set them for the U-axis only.)
- Bit 3 of parameter No. 2003 = 1 (P-I control)
- Bit 1 of parameter No. 2005 = 1 (feed forward enable)
- Parameter No. 2068 = 10000 (feed forward coefficient)
- Parameter No. 2092 = 10000 (look-ahead feed forward coefficient)

Step 4.
Suppressing load variation
Increase the value of parameter No. 2021 (within the range in which the motor does not oscillate).
Set this parameter to the value obtained from the following:
256 × (machine load inertia) / (motor rotor inertia)

For details of parameter setting, refer to the subsection titled “Feed-forward Function” in “CONTOUR ERROR SUPPRESSION FUNCTION” in “FANUC AC SERVO MOTOR α*i*/β*i* series, FANUC LINEAR MOTOR L*i*S series, FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR D*i*S series PARAMETER MANUAL (B-65270EN)”.

(11) Set bit 0 (NDPx) of parameter No.3115 to 1 for the U axis control dummy axis to disable current position display.
(12) When the U-axis control mode is released, when the tool is moved along the U-axis to the position where the reference counter is 0, set bit 3 (UOC) is parameter No.7704 to 1.

- Notes related to alarms

(1) Servo alarm, spindle alarm, and emergency stop
All axis motors are de-energized, U-axis synchronization is canceled, and the EGB synchronization mode confirmation signal EGBSM<Fn082.6> becomes “0”. If the spindle is rotating, the cancellation of synchronization may cause the tool to abruptly move along the U-axis.

(2) U-axis servo off
The U-axis motor is de-energized, but U-axis synchronization is not canceled. If the spindle is rotating, the tool may abruptly move along the U-axis since the U-axis motor is de-energized. If the tool moves long the U-axis when the U-axis is in the servo off state, the pulses corresponding to the amount of travel is stored in the servo error counter regardless of the follow-up setting by bit 0 (FUP) of parameter No. 1819 and the signal *FLWU <Gn007.5>. Therefore, when the servo off signal becomes “0”, the tool moves the along the U-axis so that the stored pulses become 0. When the EGB synchronization mode selection signal EGBS<Gn067.4> is set to “0”, the error amount also becomes 0.

(3) Other alarms
The tool stops moving along each axis with the synchronization maintained.

- Notes for safety measures

(1) Synchronization is not maintained when the U-axis is in any of the following states:
- Emergency stop (both the CNC and serial spindle)
- Servo alarm
- Spindle alarm
- Servo-off
- Control axis detach
- Power failure

If the spindle is rotating due to inertia in any of these states, the tool may abruptly move along the U-axis. In such cases, therefore, it is necessary to install some safety measures on the machine, such as applying a brake to the spindle or disengaging the clutch of the U-axis drive mechanism.

(2) Design the machine such that, when the U-axis motor comes to a halt, the tool always moves along the U-axis away from the workpiece, that is, to the safer side.

(3) When synchronization is off, it is dangerous, because the tool moves along the U-axis when the spindle rotates. The PMC must be equipped with some safety measures, such as monitoring the EGB synchronization mode confirmation signal EGBSM<Fn082.6> and preventing the spindle from rotating if the confirmation signal is “0”.

(4) When you use U-axis control by Electronic Gear Box(FSSB type), set “1” to EGB synchronization mode selection signal EGBS<Gn067.4> according to following procedure (a) to (c).
(a) Turn on CNC and confirm the ready of all spindle (All-spindle operation ready signal SRSRDY<Fn034.7>=”1”).
(b) Wait about 40msec after (a).
(c) Set the signal EGBS<Gn067.4> to “1”.
If EGBS<Gn067.4> has been “1” since turning on CNC, a shock might occur at the point where the spindle passes the first one rotation signal.

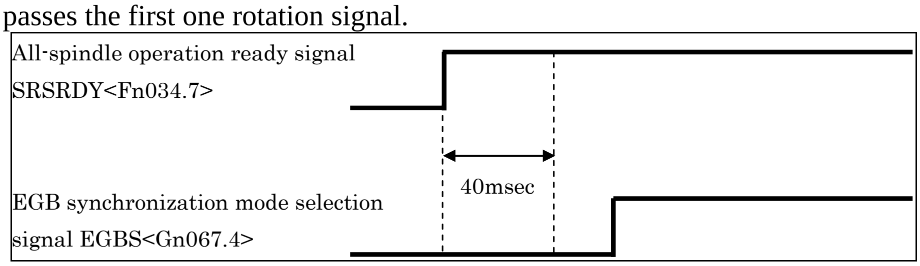


Fig. 1.9.6 (d) EGBS ON

▪ Notes

(1) A synchronization coefficient is internally represented using a fraction (K2/K1) to eliminate an error. The formula below is used for calculation.

$$\text{Synchronization coefficient} = \frac{K2}{K1} = \frac{\beta}{\alpha}$$

where
α: Number of position detector pulses per rotation about the spindle (parameter No. 7772)
β: Amount of travel by the U-axis motor per rotation about the spindle (parameter No. 7773)
In the formula above, K2/K1 is obtained by reducing β/α to lowest terms, but K1 and K2 must satisfy the following restriction:
$-2147483648 \le K2 \le 2147483647$
$1 \le K1 \le 2147483647$
When this restriction is not satisfied, alarm PS1596,“EGB OVERFLOW” is issued when synchronization starts.

(2) During synchronization, G00, G53, and canned cycles for drilling can be specified.

(3) Synchronization is maintained even if the U-axis or CNC unit is in any of the following states:
- Interlock
- Feed hold
- OT alarm
- PS alarm

(4) Synchronization pulses do not cause the machine, absolute, or relative coordinate of the U-axis to be updated. And the absolute coordinate of the U-axis is not updated when synchronization is canceled.

(5) EGB synchronization requires axis control circuits for two axes (U-axis and dummy axis). It acquires the pulses necessary for synchronization from the separate feedback connector on the dummy axis side.

(6) Any absolute position detector cannot be used for the U-axis.

(7) Controlled axes detach cannot be applied to the U-axis. If controlled axes detach is used for the U-axis, the synchronization mode is canceled.

(8) To turn synchronization on or off during automatic operation, use an M code preventing buffering to control the EGB synchronization mode selection signal EGBS<Gn067.4>.

(9) Dual check safety cannot be applied to the U-axis.
Set bit 6 (DCNx) of parameter No. 1904 to 1 to disable dual check safety of the U-axis.

(10) Perform reference position return with synchronization off, when reference position return is not performed after power on.

Signal

EGB synchronization mode selection signal
EGBS<Gn067.4>

[Classification] Input signal
[Function] Selects the EGB synchronization mode for U-axis control.
[Operation] When this signal becomes “1”, the control unit operates as follows:
Starts EGB synchronization.

All-spindle operation ready signal
SRSRDY <Fn034.7>

[Classification] Output signal
[Function] This signal posts that all spindles used are ready for operation.
[Operation] This signal becomes “1” when:
All used spindles become ready for operation.
It becomes “0” when:
Some of the used spindles are not ready for operation.

EGB synchronization mode confirmation signal
EGBSM<Fn082.6>

[Classification] Output signal
[Function] Notifies the EGB synchronization for the U-axis control in active.
[Operation] This signal becomes “1” when:
EGB synchronization in U-axis control is active.
It becomes “0” when:
EGB synchronization in U-axis control is canceled.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn067				EGBS				
Fn034	SRSRDY							
Fn082		EGBSM						

Parameter

1023	Number of the servo axis for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 80

This parameter associates each control axis with a specific servo axis.
Usually set to same number as the control axis number.
Specify values 1+8n, 2+8n, 3+8n, 4+8n, 5+8n, and 6+8n (n = 0, 1, 2, …, 9) like 1, 2, 3, 4, 5, …, 77, and 78.
For electronic gear box (EGB) controlled axes, two axes need to be specified as one pair. So, make a setting as described below.
For a slave axis, set an odd (1, 3, 5, 7, 9, ...) servo axis number. For a dummy axis to be paired, set a value obtained by adding 1 to the value set for the slave axis.

1320	Coordinate value I of stored stroke check 1 in the positive direction on each axis
1321	Coordinate value I of stored stroke check 1 in the negative direction on each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the coordinate value of stored stroke check 1 on each axis in the + or - direction in the machine coordinate system.

NOTE
1. Specify diameter values for any axes for which diameter programming is specified.
2. The area outside the area set by parameters Nos. 1320 and 1321 is a prohibited area.

	#7	#6	#5	#4	#3	#2	#1	#0
1819								**FUPx**

[Input type] Parameter input
[Data type] Bit axis

#0 FUPx To perform follow-up when the servo is off is set for each axis.
0: The follow-up signal, *FLWU<Gn007.5>, determines whether follow-up is performed or not.
When *FLWU<Gn007.5> is “0”, follow-up is performed.
When *FLWU<Gn007.5> is “1”, follow-up is not performed.
1: Follow-up is not performed.

NOTE
When using the index table indexing function, set FUPx to 1 for a control axis subject to index table indexing.

1821	**Reference counter size for each axis**

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999
Set a reference counter size.
As a reference counter size, specify a grid interval for reference position return based on the grid method.
When a value less than 0 is set, the specification of 10000 is assumed.
When a linear scale with absolute address reference marks is used, set the interval of mark 1.

1826	**In-position width for each axis**

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
The in-position width is set for each axis.
When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position. (The machine is in the in-position state.)

1828	Positioning deviation limit for each axis in movement

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.
If the positioning deviation exceeds the positioning deviation limit during movement, a servo alarm SV0411, "EXCESS ERROR (MOVING)" is generated, and operation is stopped immediately (as in emergency stop).
Generally, set the positioning deviation for rapid traverse plus some margin in this parameter.

1829	Positioning deviation limit for each axis in the stopped state

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in the stopped state for each axis.
If, in the stopped state, the positioning deviation exceeds the positioning deviation limit set for stopped state, a servo alarm SV0410, "EXCESS ERROR (STOP)" is generated, and operation is stopped immediately (as in emergency stop).

	#7	#6	#5	#4	#3	#2	#1	#0
2011								SYNx

[Input type] Parameter input
[Data type] Bit axis

#0 SYNx When the electronic gear box function (EGB) is used, this bit sets the axis to be synchronized.
0: Axis not synchronized by EGB
1: Axis synchronized by EGB
Set 1 for both of the slave and dummy axes of EGB.

> **NOTE**
> The setting of this parameter becomes valid after the power is turned off then back on.

	#7	#6	#5	#4	#3	#2	#1	#0
3115								NDPx

[Input type] Parameter input
[Data type] Bit axis

#0 NDPx The current position is:
0: Displayed.
1: Not displayed.

> **NOTE**
> When using the electronic gear box (EGB) function, set 1 for the EGB dummy axis to disable current position display.

	#7	#6	#5	#4	#3	#2	#1	#0
7700								HBR

[Input type] Parameter input
[Data type] Bit path

#0 HBR Performing a reset:
0: Cancels the EGB synchronization mode.
1: Does not cancel the EGB synchronization mode.

> **NOTE**
> To perform U-axis control, set this parameter to 1 so that performing a reset does not cancel the EGB synchronization mode.

	#7	#6	#5	#4	#3	#2	#1	#0
7702							UAX	

[Input type] Parameter input
[Data type] Bit path

#1 UAX U-axis control is:
0: Not performed.
1: Performed.

	#7	#6	#5	#4	#3	#2	#1	#0
7704					UOC			

[Input type] Parameter input
[Data type] Bit path

#3 UOC When the U-axis control mode is released, the tool is:
0: Not moved along the U-axis to the position where the reference counter is 0.
1: Moved along the U-axis to the position where the reference counter is 0.

Use this parameter to change the U-axis mode.

> **NOTE**
> Before changing the mode, be sure to perform reference position return along the U-axis and spindle orientation to change the mode at the same position (origin along the U-axis).

7772	Number of position detector pulses per rotation about the spindle

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[Valid data range] 1 to 999999999

This parameter sets the number of pulses per rotation about the spindle (master axis), for the position detector.
For an A/B phase detector, set this parameter with four pulses equaling one A/B phase cycle.

7773	Amount of travel by the U-axis motor per rotation about the spindle

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[Valid data range] 1 to 999999999

This parameter sets the amount of travel by the U-axis motor per rotation about the spindle in detection units.
When the EGB synchronization mode selection signal becomes 1, synchronization between the spindle and U-axis starts with the synchronization coefficient specified by parameters Nos. 7772 and 7773.

	#7	#6	#5	#4	#3	#2	#1	#0
7786							UFF	

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 UFF During U-axis synchronization, a interpolation command to between the U-axis and the other axes is
0: not available.
1: available.

> **NOTE**
> Set this parameter to 1, when a command like this is specified.
> Example) Axis Configuration: U(U-axis) Z(not U-axis)
> G01 U_ Z_ F_;

Alarm and message

Number	Message	Description
PS1593	EGB PARAMETER SETTING ERROR	Error in setting a parameter related to the EGB (1) The setting of bit 0 (SYNx) of parameter No. 2011 is not correct. (2) Number of pulses per rotation (parameter No. 7772 or 7773) is not set.
PS1596	EGB OVERFLOW	An overflow occurred in the calculation of the synchronization coefficient.

1.9.7 Signal-based Servo EGB Synchronous Control

Overview

This function can use input signals to make the spindle (master axis) synchronize with the servo motor (slave axis). It is possible to make the servo motor synchronize with the spindle without using programmed commands. An example of using this function might be rotary guide bushing control between the servo motor and spindle.

> **NOTE**
> 1 Using this function does not requires the electronic gear box option. When the option of the electronic gear box is effective, this function cannot be used.

Because the electronic gear box (hereafter called the EGB function), which uses digital servo for direct control, is used as a method for synchronization between the master and slave axes, the slave axis can follow the speed change of the master axis, thus realizing high-precision machining.
Using bit 0 (SVE) of parameter No. 7786 can select whether to enable/disable this function.
The PMC input signal is used to turn on/off the EGB synchronous mode.

- Example of controlled axis configuration

Spindle : EGB master axis
1st axis : X axis
2nd axis : Z axis
3rd axis : A axis (EGB slave axis)
4th aFxis : A axis (EGB dummy axis[*1] : Cannot be used as a normal controlled axis.)

(*1) One servo axis is exclusively used for the servo to directly read information about the rotation position of the master axis connected to the spindle motor.

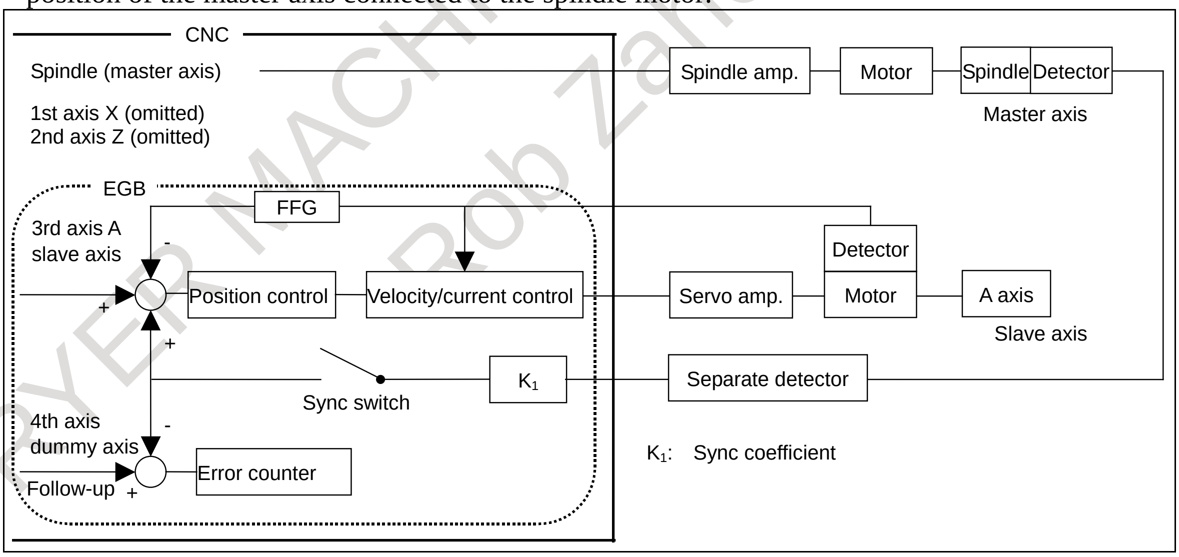


Fig. 1.9.7 (a)

For EGB axis configuration parameter setting examples, see the section on "FSSB setting".

Explanation

- Start of synchronization

Setting the EGB synchronization start signal EGBS1 to EGBS8<Gn530> to “1” starts synchronization.

- Cancellation of synchronization

Setting the EGB synchronization start signal EGBS1 to EGBS8<Gn530> to “0” cancels synchronization.

> **NOTE**
> Each of the following states cancels synchronization. When the state is cleared, synchronization restarts.
> (1) Emergency stop
> (2) Servo alarm

- Synchronization coefficient

The ratio of the slave axis travel distance to the master axis travel distance (synchronization coefficient) is converted to a detection unit ratio within the NC. If the resulting detection unit ratio exceeds the data range held in the NC, no synchronization can be attained normally, leading to alarm PS1596,"EGB OVERFLOW".
Let the synchronization coefficient be K. K in internal data form can be represented as Kn/Kd with a fraction reduced to its lowest term where Kn is the slave axis travel distance in detection unit and Kd is the master axis travel distance in detection unit.

$$K = \frac{Kn}{Kd} = \frac{\text{Slave axis travel distance in detection unit}}{\text{Master axis travel distance in detection unit}} = \frac{L}{T} \times \frac{\beta}{\alpha}$$

Where,

L : Synchronication ratio numerator (parameter No. 7784)
T : Synchronication ratio denominator (parameter No. 7785)
α : Number of position detector pulses per master rotation (parameter No. 7782)
β : Number of position detector pulses per slave rotation (parameter No. 7783)

Kn and Kd must be in the following respective ranges.

$-2147483648 \le Kn \le 2147483647$
$1 \le Kd \le 2147483647$

If Kn and Kd get out of the respective ranges, alarm PS1596 is issued.

- How to reduce the synchronous error

When you use the Electronic gear box function, to reduce the synchronous error, please apply feed-forward to the slave axis and set 100% to the parameter of feed-forward coefficient.
And please confirm the effectiveness of feed-forward by the following procedure.

[Procedure]
1. When the slave axis synchronizes only with the command from master axis, the position error of slave axis is regarded as the synchronous error. Please check that the position error (diagnosis data No.300) of the slave axis becomes 0 or so.
2. And also please check the position error is near 0 even when the speed of the master axis is changed.

Please set the following parameters to use Feed-forward function with 100% coefficient.

[Setting parameters]

Bit 3 (PIEN) of parameter No. 2003 = 1 (Slave axis)	Use PI control in velocity control
Bit 1 (FEED) of parameter No. 2005 = 1 (Slave axis)	Use Feed-forward function
Bit 1 (FFAL) of parameter No. 2011 = 1 (Slave axis)	Use Feed-forward function irrespective of feed mode
Parameter No.2068 (FF coefficient) = 10000 (Slave axis)	Feed-forward coefficient is 100%.

Please refer to the chapter of “Feed-forward Function” in FANUC AC SERVO MOTOR α*i* series FANUC AC SERVO MOTOR β*i* series FANUC LINEAR MOTOR L*i*S series FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR D*i*S series Parameter manual (B-65270EN) about the detail of Feed-forward function.

⚠ CAUTION

1. Reset, feed hold, interlock, and machine lock are invalid to a slave axis in EGB synchronization.
2. It is not possible to use controlled axis detach for a master or slave axis.

NOTE

1. When starting/canceling EGB synchronization, keep the master and slave axes at halt with the axes energized. To put it another way, make the master axis start rotating when the EGB mode confirmation signal EGBM1 to EGBM8<Fn208> is “1”. No synchronization can be attained normally if the master axis starts rotating before the EGB mode confirmation signal EGBM1 to EGBM8 becomes “1”.
2. When you use Servo EGB synchronization by Electronic Gear Box(FSSB type), set “1” to EGB synchronization start signals EGBS1-EGBS8<Gn530> according to following procedure (a) to (c).
   (a) Turn on CNC and confirm the ready of all spindle (All-spindle operation ready signal SRSRDY<Fn034.7>=”1”).
   (b) Wait about 40msec after (a).
   (c) Set the signal EGBS1-EGBS8 to “1”.
   If EGBS1-EGBS8 have been “1” since turning on CNC, a shock might occur at the point where the spindle passes the first one rotation signal.

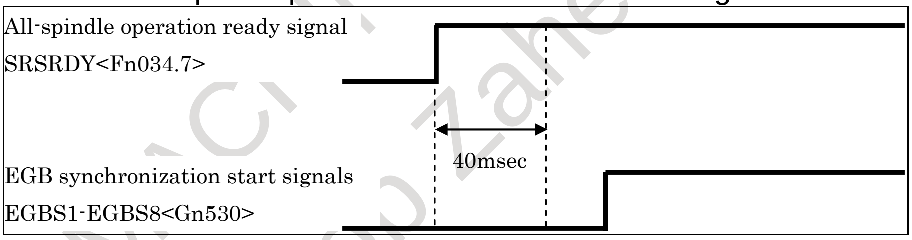


Fig.1.9.7 (b) EGBS1-EGBS8 ON

3. The rotation direction of the slave axis depends on that of the master axis. When the rotation direction of the master axis is positive, that of the slave axis is also positive. When the rotation direction of the master axis is negative, that of the slave axis is also negative. However, using the sign of the setting of parameter No. 7784 can reverse the rotation relation between the axes.
4. No synchronization can be maintained if the slave axis is in the servo-off state.
5. Actual cutting feedrate display does not take synchronization pulses into consideration.
6. In synchronization mode, it is not possible to specify G27, G28, G29, G30, and G53 for a slave axis.
7. The position of the slave axis is updated according to synchronization pulses as described below.
   - During synchronization, only the machine coordinates are updated. No absolute or relative coordinate is updated.
   - When synchronization is canceled, the distance traveled before cancellation is rounded up to 360 degrees before being added to the absolute coordinates

> **NOTE**
> 8 During EGB synchronization, no synchronization coefficient can be changed. Before changing the synchronization coefficient, cancel synchronization.
> 9 For an EGB slave axis, synchronous and composite control cannot be executed.
> 10 Using this function requires the separate detector interface unit and dedicated cables. For explanations about the cables, refer to "αi SP Series Connection Diagram" and "Details of cable K36" in "FANUC SERVO AMPLIFIER αi–B series Descriptions" (B-65412EN).

Signal

EGB synchronization start signals
EGBS1<Gn530.0>, EGBS2<Gn530.1>, …

[Classification] Input signal

[Function] Performs servo EGB synchronous control with the axis of interest used as a salve axis.

[Operation] When this signal becomes "1", the control unit operates as follows:

- Starts servo EGB synchronous control with the axis of interest used as a salve axis.

When this signal becomes "0", the control unit operates as follows:

- Cancels servo EGB synchronous control for the axis of interest used as a salve axis.

> **NOTE**
> If the master axis (spindle) or slave axis (servo axis) is not at halt, turning the EGB mode on/off causes the salve axis to accelerate/decelerate abruptly. Before turning the EGB mode on/off, be sure to keep both the master and slave axes at halt.

All-spindle operation ready signal
SRSRDY <Fn034.7>

[Classification] Output signal

[Function] This signal posts that all spindles used are ready for operation.

[Operation] This signal becomes "1" when:

All used spindles become ready for operation.

It becomes "0" when:

Some of the used spindles are not ready for operation.

EGB mode confirmation signals
EGBM1<Fn208.0>, EGBM2<Fn208.1>, …

[Classification] Output signal

[Function] Informs that EGB-based synchronization is in progress. Output is directed to the slave axis.

EGBMx

x : 1 ..... The first axis is in synchronization based on EGB.
2 ..... The second axis is in synchronization based on EGB.
3 ..... The third axis is in synchronization based on EGB.
: :
: :

[Operation] This signal becomes "1" if:

- EGB-based synchronization is in progress.

This signal becomes "0" if:

- EGB-based synchronization is canceled.

Signal Address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn530	EGBS8	EGBS7	EGBS6	EGBS5	EGBS4	EGBS3	EGBS2	EGBS1
Fn034	SRSRDY							
Fn208	EGBM8	EGBM7	EGBM6	EGBM5	EGBM4	EGBM3	EGBM2	EGBM1

Parameter

The table below gives parameters related to this function.

Parameter number	Description
1006#0,#1	An EGB slave axis and an EGB dummy axis require that the setting of a rotary axis (type A) (bit 0 (ROT) of parameter No. 1006 be 1 and bit 1 (ROS) of parameter No. 1006 be 0.
1023	Set from the FSSB setting screen. For FSSB manual setting, be sure to set the EGB axis as described below: The slave axis must be set with an odd number, and the dummy axis with an even number. They must be consecutive. Example: If the servo axis number of the slave axis is 1, the servo axis number of the dummy axis must be set to 2. If the servo axis number of the slave axis is 3, the servo axis number of the dummy axis must be set to 4.
2011#0	Specify an axis to be synchronized. Specify 1 for both an EGB slave axis and EGB dummy axis.
3115#0	The current position is not indicated for an axis for which this parameter is set to 1.Since the current position for an EGB dummy axis has no meaning, set this parameter to 1 to delete the current position indication for the axis from the screen.
7782	Pulse count of position detector per rotation about EGB master axis
7783	Pulse count of position detector per rotation about EGB slave axis
7784	Numerator of a signal-based servo EGB synchronization ratio
7785	Denominator of a signal-based servo EGB synchronization ratio
7786#0	The signal-based servo EGB synchronization function is disabled(0) / enabled(1).

For FSSB settings, see the section on “FSSB settings”.
If FSSB setting mode is automatic setting mode, setting is made automatically by inputting data to the FSSB setting screen. For the slave/dummy axes of EGB, set the value in the “M/S” item in the FSSB axis setting screen same way of the tandem setting.

Note the following points when specifying parameters for this function.

1 Specify an axis that is not used or the same name as that for a slave axis for the name of a dummy axis. Do not use a name which is usually not allowed to be used as an axis address, such as D.
2 Specify the same values for an EGB slave axis and an EGB dummy axis in the following parameters.
   1013#0 to 3 Increment system
   1004#7 Ten times minimum input increment
   1006#0,1 Rotary axis setting
   1006#3 Diameter/radius specification
   1420 Rapid traverse rate
   1421 Rapid-traverse override F0 speed
   1820 Command multiplication
   2000 and over Parameters related to digital servo
3 Specify the amount of travel per rotation about a rotation axis for a slave axis and dummy axis in a parameter No. 1260.
4 Make the specification for a dummy axis in the following way.
   1815#1 Whether to use separate detectors. Although an EGB dummy axis uses the interface of a separate detector, set these parameters to 0.
5 Reducing synchronous errors requires enabling the feed-forward function for the slave axis.
   For details, see “How to reduce the synchronous error” in “Explanation” of this chapter.

[Example 1] When the EGB master axis is the spindle and the EGB slave axis is the A-axis

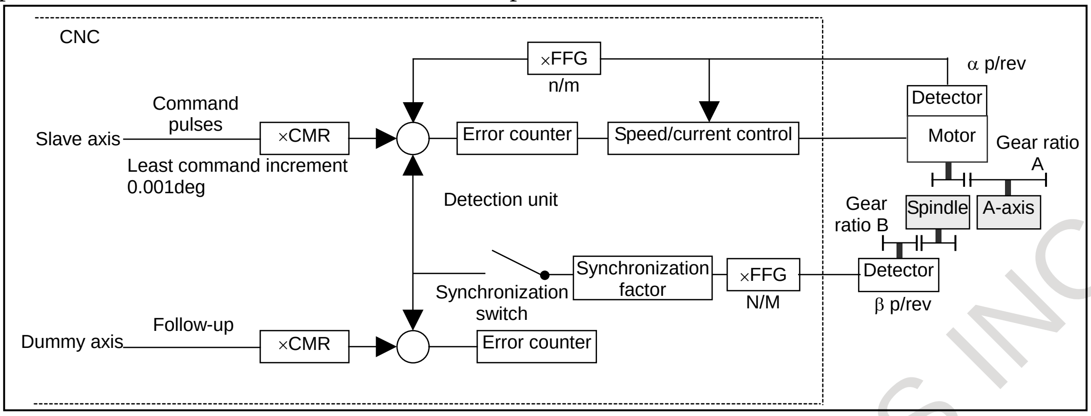


Fig.1.9.7 (c)

Gear ratio of the spindle to the detector B	: 1/1 (The spindle and detector are directly connected to each other.)
Number of detector pulses per spindle rotation β	: 4096 pulse/rev (Calculated for four pulses for one A/B phase cycle)
FFG N/M of the EGB dummy axis	: 1/1
Gear ratio of the A-axis A	: 1/36 (One rotation about the A-axis to 36 motor rotations)
Number of detector pulses per A-axis rotation α	: 1,000,000 pulses/rev
A-axis CMR	: 1
A-axis FFG n/m	: 1/100

In this case, the number of pulses per spindle rotation is:
4096 × 1/1 = 4096
Therefore, set 4096 for parameter No. 7782.
The number of pulses per C-axis rotation in the detection unit is:
1000000÷1/36×1/100 = 360000
Therefore, set 360000 for parameter No. 7783.

1023	Number of the servo axis for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 80

This parameter associates each control axis with a specific servo axis. Specify values 1+8n, 2+8n, 3+8n, 4+8n, 5+8n, and 6+8n (n = 0, 1, 2, …, 9) like 1, 2, 3, 4, 5, …, 77, and 78.

The control axis number is the order number that is used for setting the axis-type parameters or axis-type machine signals.

* For electronic gear box (EGB) controlled axes, two axes need to be specified as one pair. So, make a setting as described below.
EGB axis:
For a slave axis, set an odd (1, 3, 5, 7, 9, ...) servo axis number. For a dummy axis to be paired, set a value obtained by adding 1 to the value set for the slave axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx Setting linear or rotary axis.
#1 ROSx

ROSx	ROTx	Meaning
0	1	Rotary axis (A type)

	#7	#6	#5	#4	#3	#2	#1	#0
2005							FEEDx	

[Input type] Parameter input
[Data type] Bit axis

#1 FEEDx Feed-forward function is:
0: Invalid.
1: Valid.

Set 1 for the EGB slave axis.

> **NOTE**
> This parameter is Initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

	#7	#6	#5	#4	#3	#2	#1	#0
2011								SYN

[Input type] Parameter input
[Data type] Bit axis

#0 SYN When the electronic gear box function (EGB) is used, this bit sets the axis to be synchronized.
0: Axis not synchronized by EGB
1: Axis synchronized by EGB
Set 1 for both of the slave and dummy axes of EGB.

> **NOTE**
> The setting of this parameter becomes valid after the power is turned off then back on.

	#7	#6	#5	#4	#3	#2	#1	#0
3115								NDPx

[Input type] Parameter input
[Data type] Bit axis

#0 NDPx The current position is:
0: Displayed.
1: Not displayed.

> **NOTE**
> When using the electronic gear box (EGB) function, set 1 for the EGB dummy axis to disable current position display.

7782	Number of pulses from the position detector per EGB master axis rotation

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 999999999

For a slave axis, set the number of pulses generated from the position detector per EGB master axis rotation.
For an A/B phase detector, set this parameter with four pulses equaling one A/B phase cycle.

7783	Number of pulses from the position detector per EGB slave axis rotation

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 999999999

For a slave axis, set the number of pulses generated from the position detector per EGB slave axis rotation.
Set the number of pulses output by the detection unit.

7784	Numerator of a signal-based servo EGB synchronization ratio

[Input type] Parameter input
[Data type] 2-word axis
[Valid data range] -999999999 to 999999999

Set the numerator of a synchronization ratio for signal-based servo EGB synchronization.

The sign of this parameter specifies the direction in which the slave axis rotates.
When the sign is plus, the slave axis rotates in the positive direction (+ direction).
When the sign is minus, the slave axis rotates in the negative direction (- direction).

7785	Denominator of a signal-based servo EGB synchronization ratio

[Input type] Parameter input
[Data type] 2-word axis
[Valid data range] -999999999 to 999999999

Set the denominator of a synchronization ratio for signal-based servo EGB synchronization.

	#7	#6	#5	#4	#3	#2	#1	#0
7786								SVE

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 SVE Signal-based servo EGB synchronization is:
0: Disabled (servo and spindle synchronization is enabled).
1: Enabled (servo and spindle synchronization is disabled).

Alarm and message

Number	Message	Description
PS1593	EGB PARAMETER SETTING ERROR	Error in setting a parameter related to the EGB (1) The setting of bit 0 (SYNx) of parameter No. 2011, is not correct. (2) The salve axis has not been specified as a rotation axis. (Bit 0 (ROT) of parameter No. 1006). (3) Number of pulses per rotation (parameter No. 7782 or 7783) is not set. (4) No signal-based EGB synchronization ratio (parameters Nos. 7784 and 7785) has been set.
PS1595	ILL-COMMAND IN EGB MODE	During synchronization with the EGB, a command that must not be issued is issued. (1) Slave axis command using G27, G28, G29, G30, G33, G53, etc.
PS1596	EGB OVERFLOW	An overflow occurred in the calculation of the synchronization coefficient.

1.9.8 Electronic Gear Box (FSSB type)

Overview

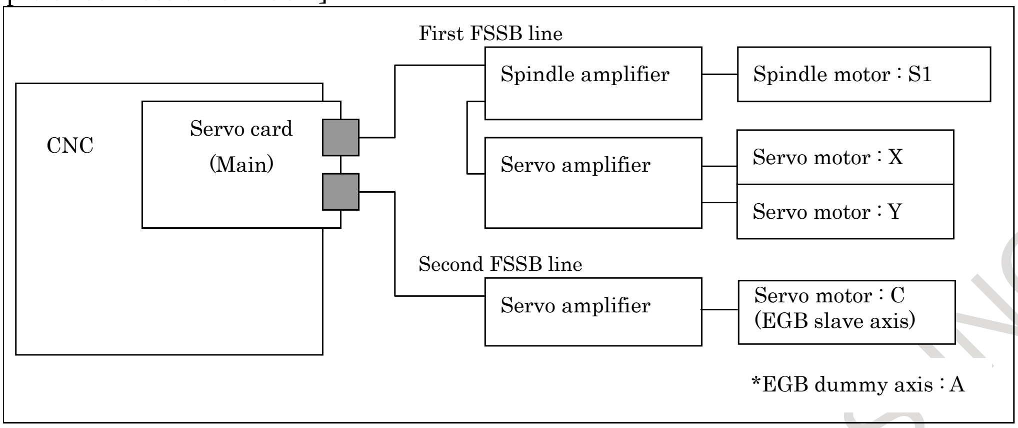
By the FSSB communication between a CNC and a spindle amplifier, rotational position information of a spindle can be transmitted from a spindle amplifier to a servo control on CNC. Electronic Gear Box (FSSB type) is a new function that a servo axis follows a spindle axis by utilizing this FSSB communication (the electronic gear box is hereinafter called the EGB). And it is a feature of this function that the separate detector interface unit becomes unnecessary compared with the ordinary function.

Explanation

The spindle amplifier and the servo amplifier used for EGB (FSSB type) need to be connected to the main servo card. The spindle amplifier and the servo amplifier used for this function can be connected to both the first FSSB line and the second FSSB line. This function can be done between two motors connected to another line.

Ex.1) EGB (FSSB type) with C axis (EGB dummy axis is A axis) and S1 spindle axis in the one path system

[Example of connection of FSSB]



[Example of parameter setting]

The following settings are necessary in addition to setting of existing parameters related to the EGB.

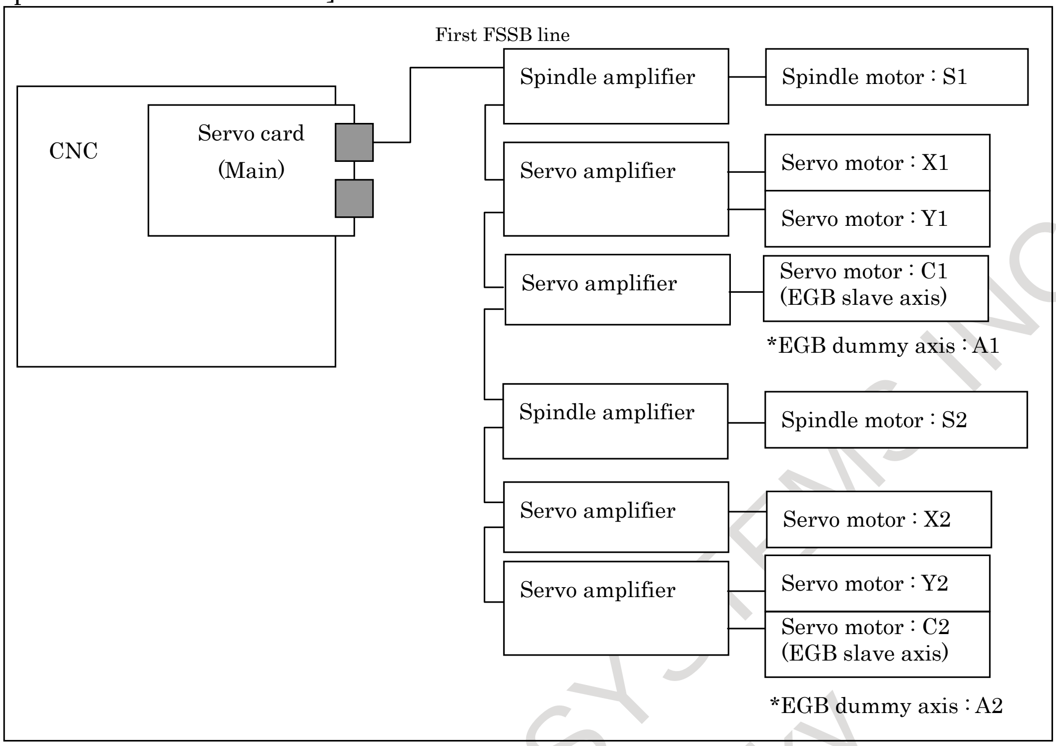
When bit 0 (FHR) of parameter No. 24203 is set to 1, position data transmission by FSSB is enabled.
And, the index number of the spindle axis that synchronizes with the EGB dummy axis used by EGB (FSSB type) is set to the parameter No. 24204 as follows.

24204	SP INDEX OF SV FSSB SYNC
X	0
Y	0
C	0
A	1

In addition, set 1 to bit 2 (FSBSYNx) of parameter No. 2429 of EGB dummy axis and bit 2 (FHESPs) of parameter No. 4549 of S1 spindle axis.

Ex.2) Two groups of EGB (FSSB type) with C1 – S1 (EGB dummy axis is A1 axis) and C2 – S2 (EGB dummy axis is A2 axis) in two paths system

[Example of connection of FSSB]



[Example of parameter setting]

The following settings are necessary in addition to setting of an existing parameters related to the EGB.

When bit 0 (FHR) of parameter No. 24203 is set to 1, position data transmission by FSSB is enabled.
And, the index number of the spindle axis that synchronizes with the EGB dummy axis used by EGB (FSSB type) is set to the parameter No. 24204 as follows.

Path 1

24204	SP INDEX OF SV FSSB SYNC
X1	0
Y1	0
C1	0
A1	1

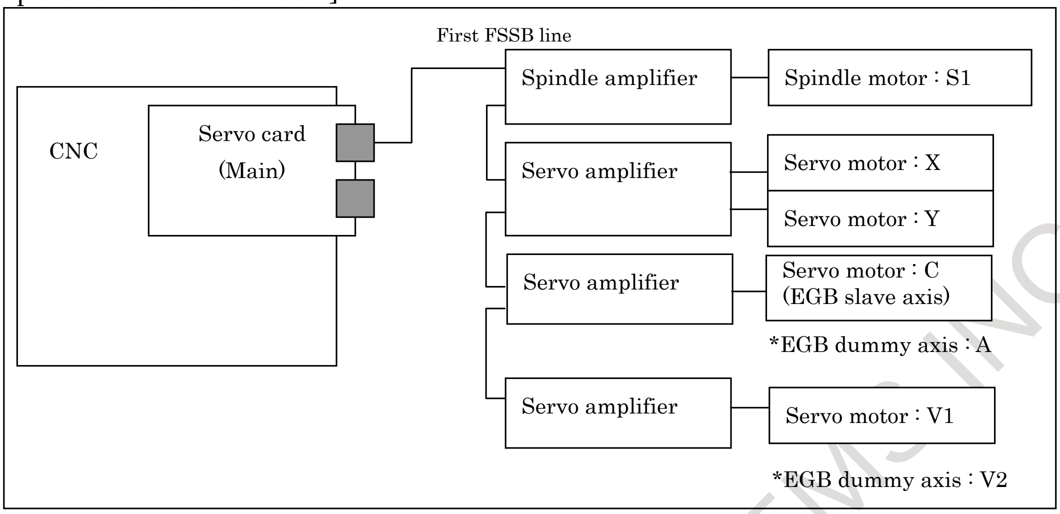
Path 2

24204	SP INDEX OF SV FSSB SYNC
X2	0
Y2	0
C2	0
A2	2

In addition, set 1 to bit 2 (FSBSYNx) of parameter No. 2429 of A1 and A2 axes and bit 2 (FHESPs) of parameter No. 4549 of S1 and S2 spindle axes.

Ex.3) EGB (FSSB type) by EGB 2 pair with using common master axis to C1 – S1 (EGB dummy axis is A axis) and V1 – S1 (EGB dummy axis is V2 axis) in the one path system

[Example of connection of FSSB]



[Example of parameter setting]

The following settings are necessary in addition to setting of an existing parameters related to the EGB.

When bit 0 (FHR) of parameter No. 24203 is set to 1, position data transmission by FSSB is enabled.
And, the index number of the spindle axis that synchronizes with the EGB dummy axis used by EGB (FSSB type) is set to the parameter No.24204 as follows.

24204	SP INDEX OF SV FSSB SYNC
X	0
Y	0
C	0
A	1
V1	0
V2	1

In addition, set 1 to bit 2 (FSBSYNx) of parameter No. 2429 of A and V2 axes and , bit 2 (FHESPs) of parameter No. 4549 of S1 spindle axis.

The servo axes that can receive the feedback data from the spindle axis through FSSB are eight axes or less in all system. If nine axes or more of the servo axis or illegal index number of the spindle axis is set, the alarm (PW0037) “SV/SP COMBINATION ERROR” is issued.

NOTE

1 This function can be used by the following EGB functions.
 - Electronic gear box
 - U-axis control
  - Signal-based servo EGB synchronous control

2 As following EGB functions are realized by 2 spindles, it is impossible to apply this FSSB type.
 - Spindle electronic gear box
 - Simple spindle electronic gear box

NOTE

3 When the parameter is changed, the power must be turned off before an operation is continued.

4 EGB (FSSB type) cannot be used with the analog spindle or the spindle control with servo motor. If the above EGB (FSSB type) is set, the alarm (PW0037) “SV/SP COMBINATION ERROR” is issued.

5 When you don’t use spindle sensor and drive the axis only by motor a sensor, arbitrary gear ratio between motor and spindle (parameter No.4171-No.4174) is used. In the case, please set 1 in bit 1 (EGBAGR) of parameter No.4004.

6 When you use automatic phase synchronization of EGB (FSSB type) from the stop state or slow moving, you have to rotate the spindle axis at least one rotation with higher speed than 10min^{-1} before the phase synchronization operation.

7 With EGB dummy axis used by EGB (FSSB type), the following functions cannot be used.
- FSSB high-speed rigid tapping
- Servo/spindle synchronous control
- Fully closed control

If EGB (FSSB type) is used with either of the above functions, the alarm (SV0417) “ILL DGTL SERVO PARAMETER” is issued.

8 When you use U-axis Control or Signal-based Servo EGB Synchronous Control by Electronic Gear Box(FSSB type), set “1” to EGB synchronization mode selection signal EGBS<Gn067.4> or EGB synchronization start signals EGBS1-EGBS8<Gn530> according to following procedure (a) to (c).

(a) Turn on CNC and confirm the ready of all spindle (All-spindle operation ready signal SRSRDY<Fn034.7>=”1”).

(b) Wait about 40msec after (a).

(c) Set the signal EGBS or EGBS1-EGBS8 to “1”.

If EGBS<Gn067.4> or EGBS1-EGBS8<Gn530> has been “1” since turning on CNC, a shock might occur at the point where the spindle passes the first one rotation signal.

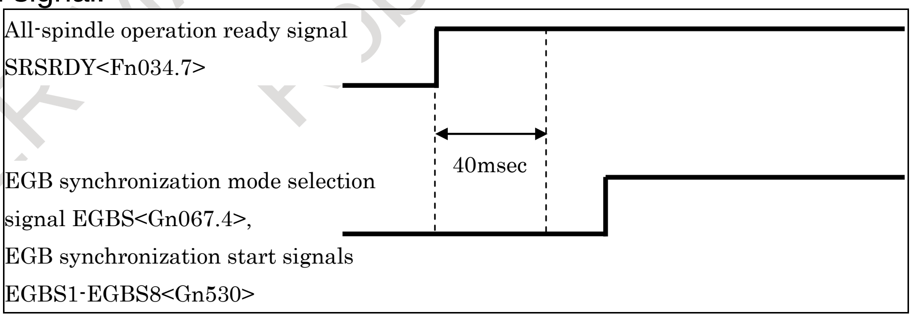


Fig.1.9.8 (a) EGBS or EGBS1-EGBS8 ON

Signal

EGB synchronization mode selection signal
EGBS<Gn067.4>

[Classification] Input signal

[Function] Selects the EGB synchronization mode for U-axis control.

[Operation] When this signal becomes “1”, the control unit operates as follows:
Starts EGB synchronization.

EGB synchronization start signals
EGBS1<Gn530.0>, EGBS2<Gn530.1>, …

[Classification] Input signal

[Function] Performs servo EGB synchronous control with the axis of interest used as a slave axis.

[Operation] When this signal becomes “1”, the control unit operates as follows:
- Starts servo EGB synchronous control with the axis of interest used as a slave axis.

When this signal becomes “0”, the control unit operates as follows:
- Cancels servo EGB synchronous control for the axis of interest used as a slave axis.

> **NOTE**
> If the master axis (spindle) or slave axis (servo axis) is not at halt, turning the EGB mode on/off causes the slave axis to accelerate/decelerate abruptly. Before turning the EGB mode on/off, be sure to keep both the master and slave axes at halt.

All-spindle operation ready signal
SRSRDY <Fn034.7>

[Classification] Output signal

[Function] This signal posts that all spindles used are ready for operation.

[Operation] This signal becomes “1” when:
All used spindles become ready for operation.

It becomes “0” when:
Some of the used spindles are not ready for operation.

Signal Address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn067				EGBS				
Gn530	EGBS8	EGBS7	EGBS6	EGBS5	EGBS4	EGBS3	EGBS2	EGBS1
Fn034	SRSRDY							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
2429						FSBSYNx		

[Input type] Parameter input

[Data type] Bit axis

#2 FSBSYNx Servo control by EGB (FSSB type) is:
0: Disabled.
1: Enabled.

> **NOTE**
> 1 When using EGB (FSSB type), set this parameter to 1.
> 2 In the EGB dummy axis, in which bit 2 (FSBSYNx) of parameter No.2429 is 1, the following functions cannot be used. If EGB (FSSB type) is used with either of the following functions, the alarm (SV0417) “ILL DGTL SERVO PARAMETER” is issued. (Detail number of diagnosis data No.0352 :4292 or 4291)
>   - FSSB high-speed rigid tapping (Bit 1 (FHRSVx) of parameter No.2429)
>   - Servo/spindle synchronous control (Bit 4 (SPSx) of parameter No.2016)
>
> 3 When this parameter is set, the power must be turned off before operation is continued.

	#7	#6	#5	#4	#3	#2	#1	#0
4004							EGBAGR	

[Input type] Parameter input
[Data type] Bit spindle

#1 EGBAGR Arbitrary gear ratio between the motor and spindle (parameter No.4171-No.4174) by EGB (FSSB type) is:
0: Disabled.
1: Enabled.

> **NOTE**
> 1 To use arbitrary gear ratio between the motor and spindle in case of using a motor sensor only without a spindle sensor, set the parameter EGBAGR to 1.
>   For details of parameter of arbitrary gear ratio (No.4171-No.4174), refer to spindle parameter manual (B-65280).
> 2 To use this function bit, it’s necessary to prepare the applicable CNC software and spindle software.
> 3 When bit 1 (EGBAGR) of parameter No.4004 is 0, the feedback pulsed from EGB master axis doesn’t take the arbitrary gear ratio into account.
> 4 Do not change parameter of arbitrary gear ratio and clutch/gear signals (CTH) during EGB mode. If parameters or signals are changed, the alarm (SP9167) "Spindle control sequence fault" and the alarm (SV0385) "Serial data error (EXT)" will occur.

	#7	#6	#5	#4	#3	#2	#1	#0
4549						FHESPs		

[Input type] Parameter input
[Data type] Bit spindle

#2 FHESPs Spindle control by EGB (FSSB type) is:
0: Disabled.
1: Enabled.

NOTE
1 When using EGB (FSSB type), set this parameter to 1.
2 In the spindle axis, in which bit 2 (FHESPs) of parameter No.4549 is 1, FSSB high-speed rigid tapping (bit 1 (FHRSPs) of parameter No.4549) cannot be used. If EGB (FSSB type) is used with the FSSB high-speed rigid tapping, the alarm (SP9068) “ILLEGAL SPINDLE PARAMETER” is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
24203								FHR

NOTE
When the parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 FHR Position data transmission by FSSB is:
0: Disabled.
1: Enabled.

NOTE
1 In addition, it is necessary to set parameter No.24204.
2 When the following function is used, set 1 to this parameter.
- FSSB high-speed rigid tapping
- Electronic gear box (FSSB type)
- Servo/Spindle synchronous control (FSSB type)

24204	The index number of the spindle axis that synchronizes to each servo axis

NOTE
When the parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to the maximum number of spindles

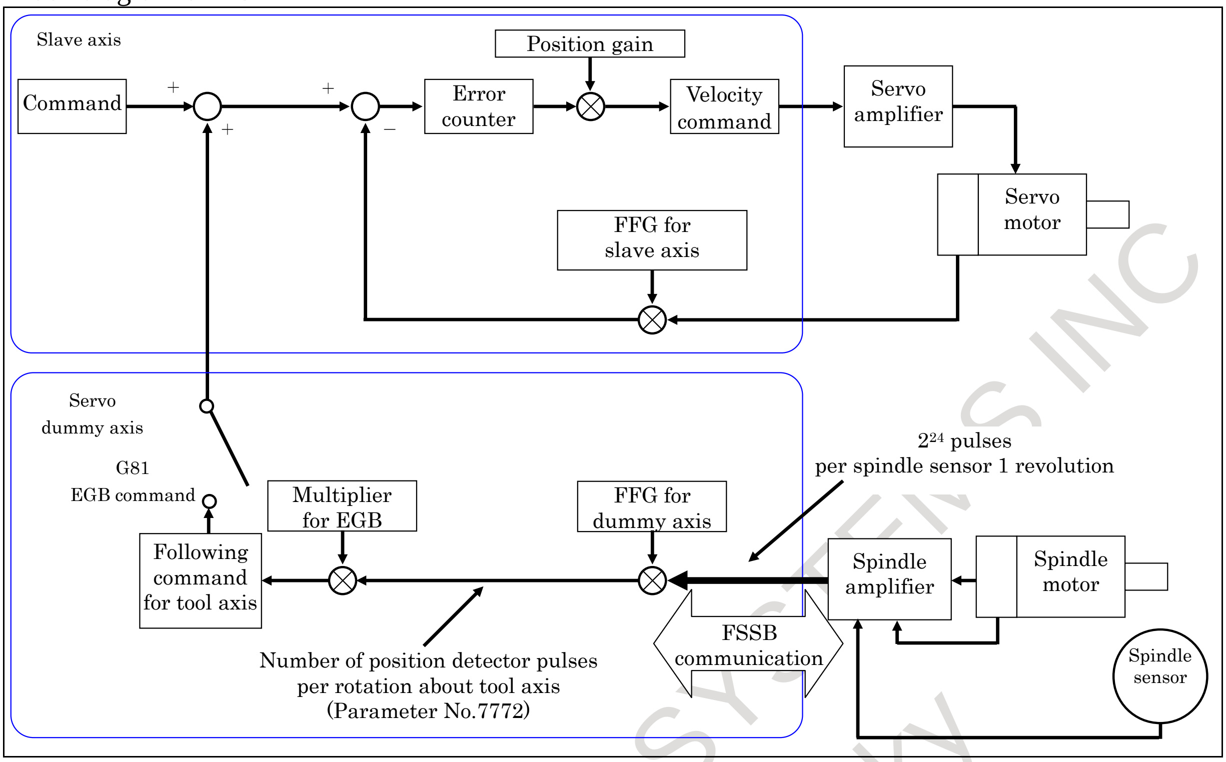
Set the index number of the spindle axis that synchronizes with EGB dummy axis used by EGB (FSSB type).

NOTE
1 When bit 0 (FHR) of parameter No.24203 is 1, this parameter is enabled.
2 Eight or less index numbers of the spindle axis can be set in the system. If nine or more index numbers of the spindle axis are set, the alarm (PW0037) "SV/SP COMBINATION ERROR" is issued.

Reference) Existing parameters related to the EGB.

Parameter number	Description
1006#0 1006#1	To specify the distance for the slave axis by L in the synchronization command, an EGB slave axis and an EGB dummy axis require that the setting of a rotary axis (type A) (bit 0 (ROT) of parameter No. 1006 be 1 and bit 1 (ROS) of parameter No. 1006 be 0.
1023	Set from the FSSB setting screen. For FSSB manual setting, be sure to set the EGB axis as described below: The slave axis must be set with an odd number, and the dummy axis with an even number. They must be consecutive. Example: If the servo axis number of the slave axis is 1, the servo axis number of the dummy axis must be set to 2. If the servo axis number of the slave axis is 3, the servo axis number of the dummy axis must be set to 4.
2011#0	Specify an axis to be synchronized. Specify 1 for both an EGB slave axis and EGB dummy axis.
2084/2085	Flexible feed gear. The feed back data from spindle position sensor is regarded as 2^{24} pulses/rev for input to dummy axis. If you want to thin it out, please use Flexible feed gear in dummy axis. (See following figure)
3115#0	The current position is not indicated for an axis for which this parameter is set to 1.Since the current position for an EGB dummy axis has no meaning, set this parameter to 1 to delete the current position indication for the axis from the screen.
7700#0	The synchronization mode is canceled (0)/not canceled (1) by a reset.
7700#2	Compensation direction for helical gear compensation
7701#3	At the start of synchronization (G81), synchronization is started (0)/not started (1) if the number of hob threads L is specified as 0.
7702#0	The specifiable number of teeth, T, at the start of synchronization (G81) is not reduced to a 1/10 of a specified value (0)/reduced (1).
7702#3	The retract function with an alarm is disabled (0)/enabled (1).
7703#0	During synchronization (G81), feed per revolution is performed for feedback pulses (0)/pulses converted to the speed for the workpiece axis(1).
7703#1,#2	Specify when to perform a retract operation with the retract function with an alarm; during synchronization; during synchronization and automatic operation; or during synchronization or automatic operation.
7709	Number of the axial feed axis in helical gear compensation
7731#0	The EGB command is G80 and G81(0)/G80.4 and G81.4(1).
7731#3	When the automatic phase synchronization function for the electronic gear box is disabled, the G81 command cannot be issued again (an alarm is issued) (0)/can be issued again (1) during EGB synchronization.
7731#5	In EGB synchronization start command G81.4, the number of teeth is specified in T (0)/specified in R (1).
7740	Feedrate during retraction
7741	Retract amount
7772	Number of position detector pulses per rotation about tool axis. Please set the value which takes the Flexible feed gear and EGB exponent setting into consideration. (See following figure.)
7773	Number of position detector pulses per rotation about workpiece axis
7782	Pulse count of position detector per rotation about EGB master axis
7783	Pulse count of position detector per rotation about EGB slave axis
7784	Numerator of a signal-based servo EGB synchronization ratio
7785	Denominator of a signal-based servo EGB synchronization ratio
7786#0	The signal-based servo EGB synchronization function is invalid (0) / valid (1).

Block diagram of EGB



Example 1)

Gear ratio of the spindle to the spindle sensor: 1/1 (The spindle and detector are directly connected to each other.)
FFG N/M of the EGB dummy axis: 1/1
Gear ratio of the C-axis (Slave axis): 1/36 (One rotation about the C-axis to 36 motor rotations)
Number of detector (αi Pulsecoder) pulses per C-axis rotation: 1,000,000 pulses/rev
C-axis CMR: 1
C-axis detection unit: 0.001 deg

In this case, the number of position detector pulses per one rotation in tool axis (spindle) is:
$2^{24} * 1/1 = 16777216$
Therefore, set 16777216 for parameter No. 7772.
The number of pulses per C-axis rotation in the detection unit is:
360(deg) / 0.001(deg) = 360000
Therefore, set 360000 for parameter No. 7773.

Regarding FFG for C-axis is
360000 / (36*1000000) = 1/100

Example 2)

Gear ratio between spindle and spindle motor: 1/3 (The spindle sensor is none. Arbitrary gear ratio : 1/3)
FFG N/M of the EGB dummy axis: 1/1
The configuration of C axis is same as example 1.

In this case, by setting 1 in bit 1 (EGBAGR) of parameter No.4004, the feedback data from spindle to servo takes arbitrary gear ratio into account. In spindle one rotation, the motor sensor rotates 3 revolutions. But as the feedback pulses from the motor sensor is multiplied by arbitrary gear ratio, the number of position detector pulses per one rotation in tool axis (spindle) is calculated by following formula,
$2^{24}*3*(1/3)*(1/1) = 16777216$

Motor 3 revolutions any gear ratio FFG. It means No.7772 should be 16777216.

Alarm and message

Number	Message	Description
PW0037	SV/SP COMBINATION ERROR	- The servo axis has dummy axis setting. Please check parameters (No.1023, bit 0 of No.2009, bit 4 of No.11802). - The setting for spindle axis number to synchronize with servo is wrong. Please check parameters (bit 0 of No.3716, No.3717, No.24204).
SV0417	ILL DGTL SERVO PARAMETER	- The setting for the function, which is not available with EGB (FSSB type) together, is done. Please turn it off.
SP9068	ILLEGAL SPINDLE PARAMETER	- EGB (FSSB type) and FSSB high-speed rigid tapping cannot be used together. Please check the parameters (bit 1 of No.4549, bit 2 of No.4549) - In the case of EGB (FSSB type) using, please turn off FSSB high-speed rigid tapping,
SP9167	SPINDLE CONTROL SEQUENCE FAULT	- Arbitrary gear ratio should not be changed during EGB mode. When you need to change arbitrary gear ratio, you have to change the parameters and clutch/gear signals before starting EGB operation.

Diagnosis data

1612	The index number of the spindle axis that synchronizes with each servo axis

[Data Type] Byte axis

The index number of the spindle axis that can use direct communication between a spindle amplifier and a servo amplifier on FSSB connection is displayed.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	Electronic Gear Box Electronic Gear Box 2 Pair U-axis Control Signal-based Servo EGB Synchronous Control
OPERATOR’S MANUAL (B-64694EN)	Electronic Gear Box Electronic Gear Box 2 Pair U-axis Control

1.10 DUAL POSITION FEEDBACK TURNING MODE / COMPENSATION CLAMP

Overview

To the axis to be controlled with the dual position feedback function, add the turning mode and a compensation clamp. For details of the dual position feedback function, refer to the FANUC AC SERVO MOTOR α*i*/β*i* series, FANUC LINEAR MOTOR L*i*S series, FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR D*i*S series PARAMETER MANUAL (B-65270EN).

- Turning mode

Select the turning mode by inputting the dual position feedback turning mode selection signal. In the turning mode, the full closed side feedback for the axis set in bit 0 (STHx) of parameter No. 11803 is ignored. If a move command is issued in the turning mode, it is operated only with control with a semi-closed loop.

This function is used to switch the control method depending on the case; for example, to perform position control on a given axis, using the servo motor or to control a given axis as a spindle, using a spindle motor.

In the turning mode, full closed side feedback is ignored, so that all coordinates (machine, workpiece, and local coordinates) will be shifted by the amount of operation in the turning mode. Thus, after the end of the turning mode, a reference position return is required to correct the coordinates. If an attempt is made to perform automatic operation without performing a reference position return, alarm PS0224 "ZERO RETURN NOT FINISHED" is issued.

> **NOTE**
> An absolute position detector cannot be used for the axis that uses the turning mode.

- Compensation clamp

The compensation operation of dual position feedback is canceled by inputting the dual position feedback compensation clamp signal. When the compensation operation is canceled, full closed side feedback is interrupted and an operation with a semi-closed loop is assumed. If a mechanical clamp is to be performed on the axis on which to perform dual position feedback, a compensation clamp must be performed. This function is enabled for an axis for which bit 1 (CDPx) of parameter No. 11803 is 1.

Signal

Dual position feedback turning mode selection signal HBTRN <Gn531.3>

[Classification] Input signal
[Function] Selects the turning mode.
[Operation] When set to “1”, this signal selects the dual position feedback turning mode, ignoring the full closed side feedback for the axis set in bit 0 (STH) of parameter No. 11803.

Dual position feedback compensation clamp signal *CL1 to *CL8<Gn548>

[Classification] Input signal
[Function] Cancels the compensation operation of dual position feedback.
[Operation] When set to “0”, this signal cancels the compensation operation of dual position feedback. This signal is valid only if bit 1 (CDPx) of parameter No. 11803 is 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn531					HBTRN			
Gn548	*CL8	*CL7	*CL6	*CL5	*CL4	*CL3	*CL2	*CL1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11803							CDPx	STHx

[Input type] Parameter input
[Data type] Bit axis

#0 STHx The dual position feedback turning mode is:
0: Disabled.
1: Enabled.

> **NOTE**
> Before the dual position feedback turning mode function can be used, a setting to enable dual position feedback is required in addition to the setting of this bit.

#1 CDPx Dual position feedback compensation clamping is:
0: Not performed.
1: Performed.

> **NOTE**
> Before the dual position feedback compensation clamp function can be used, a setting to enable dual position feedback is required in addition to the setting of this bit.

Alarm and message

Number	Message	Description
PS0224	ZERO RETURN NOT FINISHED	A reference return has not been performed before the start of automatic operation. (Only when bit 0 (ZRNx) of parameter No. 1005 is 0) Perform a reference position return.

1.11 FUNCTION OF DECELERATION STOP IN CASE OF POWER FAILURE

Overview

If a power failure occurs during an axial movement, this function stops the movement by decreasing the speed on each axis at a rate specified in parameter No. 1791. This function prevents the machine from being damaged by an overrun.

Explanation

- Deceleration pattern

Both cutting feed and rapid traverse are decelerated linearly, at a constant rate.

- Example of deceleration

If the rates on the X-axis and Y-axis are different

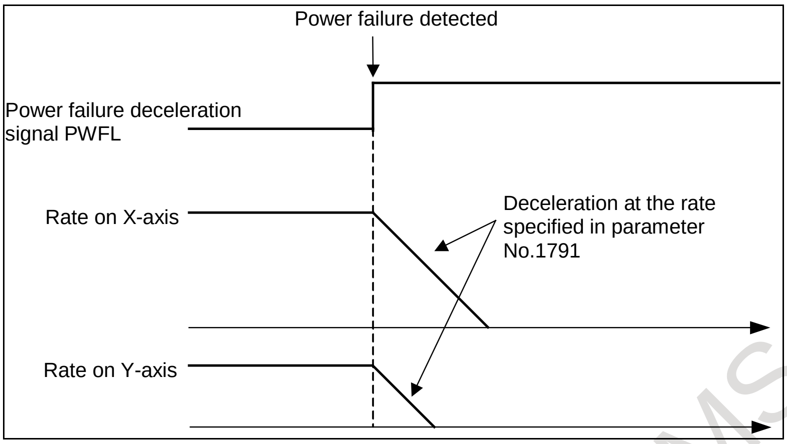


Fig. 1.11

> **NOTE**
> 1 After the completion of deceleration started by bringing the power failure deceleration signal PWFL to "1", no axial movement can be made.
> 2 To make another axial movement, turn the power off and on again.
> 3 This function does not perform deceleration on a torque control axis or a velocity control axis under PMC axis control or on an EGB axis.
> 4 Prevent an emergency stop from occurring until the deceleration is completed.
> 5 The stored stroke check function is disabled while deceleration by this function is in progress.

- Application

This function is used to decelerate and stop a linear motor in a system connected to a power-failure backup module and sub-module C (capacitor module), in case of a power failure.

- Effect of application

If a power failure occurs, energy required to decelerate and stop the linear motor is supplied from sub-module C (capacitor module). The amount of energy required for deceleration increases as the time constant decreases, or as the acceleration increases.
Generally, linear-motor machines perform high-acceleration operation, so that the time constant is set to the lowest possible level. A number of C sub-modules (capacitor modules) are required to decelerate and stop the linear motor when a power failure occurs.
The number of necessary C sub-modules (capacitor modules) can be reduced by using this function to lower the acceleration in deceleration performed in case of a power failure.

Caution

If an acceleration lower than the normal acceleration is set by this function, the braking distance at a power failure becomes longer than usual. Accordingly, a collision with a stroke end can occur, depending on the acceleration start position at a power failure.
Because the rate of the collision with a stroke end is decreased by this function, the impact of the collision is reduced than that occurring without this function (when a DB stop occurs).

Signal

Power failure deceleration signal PWFL <G203.7>

[Classification] Input signal

[Function] Indicates that a power failure has been detected.

[Operation] When this signal is set to "1", the control unit performs the following operation.

- Immediately starts deceleration at the constant rate specified in parameter No.1791, and stops the movement.

> **NOTE**
> 1 This signal is effective on all the paths.
> 2 Avoid changing any state after the signal is brought to “1”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G203	PWFL							

Parameter

1791	Acceleration rate on each axis for the outage-time deceleration stop function

[Input type] Parameter input

[Data type] Real axis

[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] Refer to the standard parameter setting table (D)
(0.0 to +100000.0 for the metric system, 0.0 to +10000.0 for the inch system)

Set an acceleration rate for deceleration on an axis on which the tool is decelerated to a stop at the time of power outage.
On an axis for which this parameter is set to 0, deceleration based on the outage-time deceleration signal is not performed.

In synchronization control or tandem control, set the same parameter for the master axis and slave axis.

1.12 FLEXIBLE SYNCHRONIZATION CONTROL

1.12.1 Flexible Synchronization Control

Overview

This function is provided for those machines like hobbing machines that require the synchronization of various multiple gear ratios.
Synchronization with this function enables up to four pairs to be operated independently and simultaneously. This achieves special functions for hobbing machines such as the synchronization of the hobbing axis and a single workpiece axis, Z-C synchronization in helical gear cutting, and Y-C synchronization in a hobbing axis shift.
Specifications for flexible synchronization control are as follows:

1) A master axis number, a slave axis number, and a gear ratio are set in parameters.
2) There can be up to four groups to these parameters. Synchronization of the four groups can be executed at the same time.
3) A single slave axis can be specified for multiple master axes.
4) Synchronization is started and canceled with DI signals from the PMC.
   If DI signals are to be switched during automatic operation, this needs to be performed with an M code set in a parameter.

5) Two Cs axes can be used as master and slave axes.
6) A retract operation can be performed during the flexible synchronization control mode and inter-path flexible synchronization control mode. The retract operation complies with the specifications of the general-purpose retract function. (Refer to section, “GENERAL PURPOSE RETRACT”, in this manual for descriptions of the specifications of the general-retract function.)

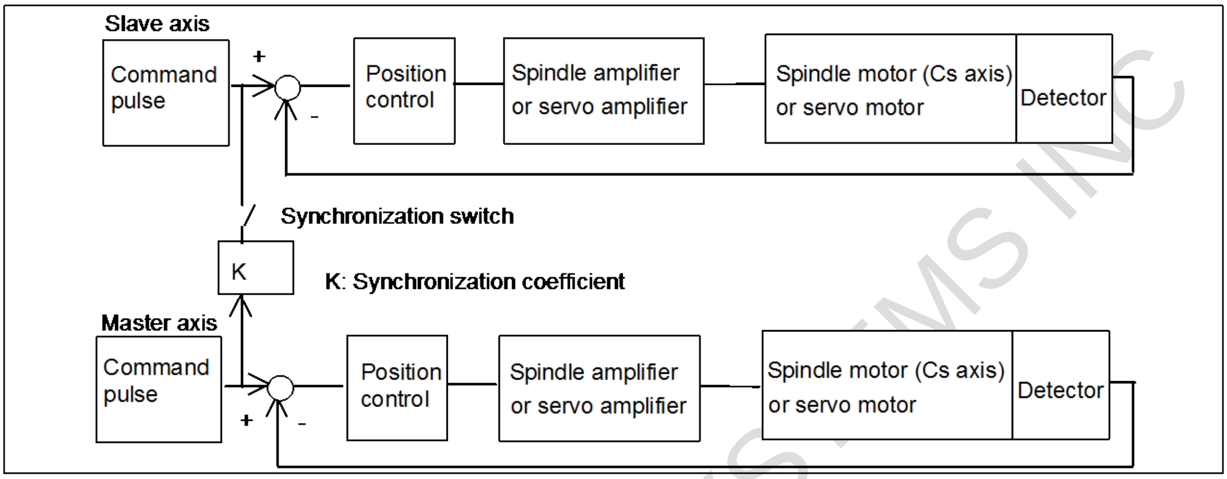


Fig. 1.12.1 Block diagram

Parameter setting

Parameters for flexible synchronization control include the following:
(1) Denominator for determining a gear ratio (parameters Nos. 5681, 5683, 5685, and 5687)
(2) Numerator for determining a gear ratio (parameters Nos. 5680, 5682, 5684, and 5686)
(3) Exponent for the denominator of a gear ratio (parameters Nos. 5690, 5691, 5692, and 5693)
(4) Master axis number (parameters Nos. 5660, 5662, 5664, and 5666)
(5) Slave axis number (parameters Nos. 5661, 5663, 5665, and 5667)
(6) M code number for turning on the synchronization mode (parameters Nos. 5670, 5672, 5674, and 5676)
(7) M code number for turning off the synchronization mode (parameters Nos. 5671, 5673, 5675, and 5677)
(8) Parameters related to the update of machine coordinates (bits 0 (ACA) to 3 (ACD) of parameter No.5668)

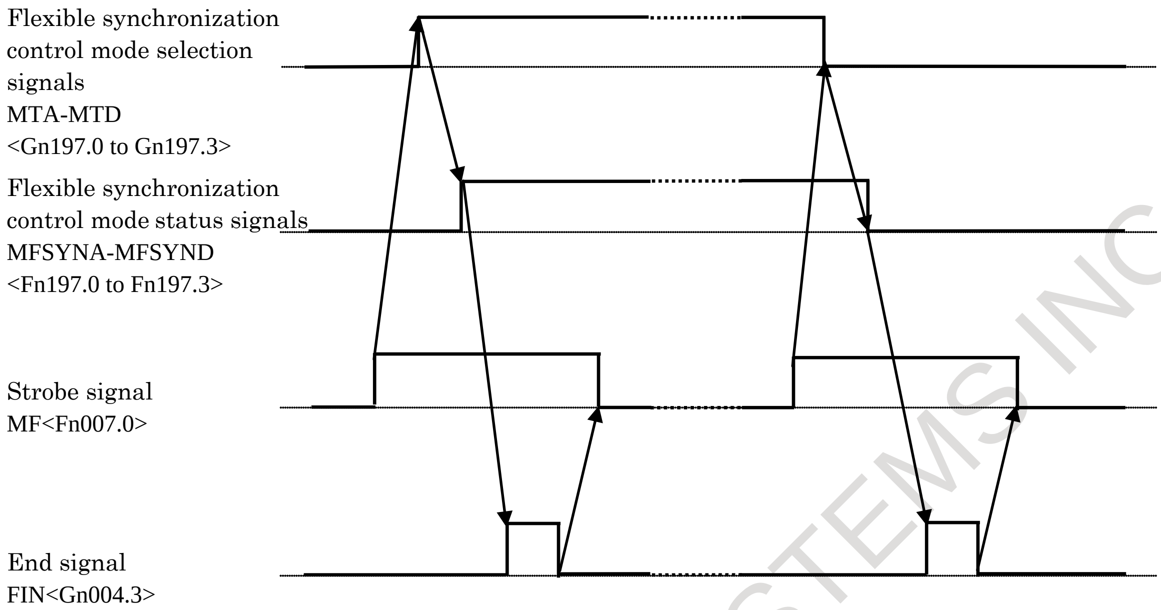
Start of synchronization

Synchronization is started by setting one of the flexible synchronization control mode selection signals MTA, MTB, MTC, and MTD<Gn197.0, Gn197.1, Gn197.2, Gn197.3> to "1". Note that to set one of MTA, MTB, MTC, and MTD to "1" during automatic operation, this needs to be performed with an M code set in parameter No. 5670, 5672, 5674, or 5676. Up to three of these M codes can be specified in a single block by enabling the multiple-Ms-per-block command (setting bit 7 (M3B) of parameter No. 3404 to 1).
When one of MTA, MTB, MTC, and MTD is accepted, one of the flexible synchronization control mode status signals MFSYNA, MFSYNB, MFSYNC, and MFSYND<Fn197.0, Fn197.1, Fn197.2, Fn197.3> is set to "1".

Cancellation of synchronization

Synchronization is canceled by setting one of the flexible synchronization control mode selection signals MTA, MTB, MTC, and MTD<Gn197.0, Gn197.1,Gn197.2,Gn197.3> to "0".
When one of MTA, MTB, MTC, and MTD is accepted, one of the flexible synchronization control mode status signals MFSYNA, MFSYNB, MFSYNC, and MFSYND<Fn197.0, Fn197.1, Fn197.2, Fn197.3> is set to "0".

Example of time chart for start and cancellation of synchronization



G27/G28/G29/G30/G53 commands

If an attempt is made to issue one of these commands for a master axis or a slave axis during flexible synchronization control, alarm PS0010 "IMPROPER G-CODE" is issued, but by setting bit 2 (FRF) of parameter No. 13421 to 1, they can be issued for a master axis.
In this case, the slave axis is linked to the operation of the master axis, so that its movement is the same as in normal flexible synchronization control.
Note, however, that even if parameter FRF is set to 1, if an attempt is made to issue G28 when a reference position has not been established for a master axis or to issue it for a slave axis, alarm PS5381 "INVALID COMMAND IN FSC MODE" is issued.

Program example

Axis configuration X, Y, Z, B (Cs axis), C, U, V
Group A: Master axis B, slave axis C, gear ratio 1:50, on M50, off M51
Group B: Master axis Z, slave axis C, gear ratio 1:5, on M52, off M53
Group C: Master axis Y, slave axis C, gear ratio 23:20, on M54, off M55
Group D: Master axis B, slave axis U, gear ratio 1:100, on M56, off M57

```
G90 G00 X111.5 Z410.0 Y75.0 B0 C0 ; ... Move to the start point
M50 ;   ....................................... Start of B - C synchronization
M52 ;       ................................... Start of Z - C synchronization
Mxx ;       ................................... Rotate about the hobbing axis
G04 X1000 ;.................................... Wait until rotation about the hobbing axis stabilizes.
G00 X71.3 ;      ............................. X-axis approach 1
G01 X61.2 F100 ;    .......................... X-axis approach 2
G01 Z369.4 F40.0 ;............................ Helical gear cutting
G00 X111.5 ;      ............................ X-axis escape
  :
M54 ;     ..................................... Start of Y - C synchronization
G91 G01 Y3.0 F100.0 ;    ..................... Y-axis shift
M55 ;   ....................................... End of Y - C synchronization
  :
G90 G00 U100.0 V200.0 B0 ; ................... Move to the dressing start point
M56 ;   ....................................... Start of B - U synchronization
Mxx ;     ..................................... Rotate about the hobbing axis
G04 X1000 ;    ............................... Wait until rotation about the hobbing axis stabilizes.
```

```
G01 V100.0 ;............................................. V-axis approach
G01 U200.0 ;    .......................................... Dressing
G00 V200.0 ;............................................. V-axis escape
M57 ;            ............................................. End of B - U synchronization
```

Flexible synchronization control in case a master axis is controlled by “Spindle Control with Servo Motor”

Flexible synchronization control can be applied to the case where an axis that is controlled by “Spindle Control with Servo Motor” is assigned as a master axis, when flexible synchronization control selection signal MTA, MTB, MTC, or MTD<Gn197.0 to Gn197.3 > is set to "1" in the SV speed control mode. In this case, a master axis should be in the SV speed control mode of “Spindle Control with Servo Motor” to enter flexible synchronization mode. Please do not change the SV speed mode during flexible synchronization control. Flexible synchronization control should be turned on/off during the SV speed mode according to the following example of programming.

Example of the machining program)

```
G00 X10.Z50.;
S0P11;
G96.4P11;       : SV speed control mode ON
M16;            : Flexible synchronization control ON
S100P11;
:
:
S0P11;
M17;            : Flexible synchronization control OFF
N9G96.1P11;     : SV speed control mode OFF
```

Retract by alarm

In case that bit 3 (ART) of parameter No. 7702=1, when a CNC alarm occurs during flexible synchronization control or automatic operation, the axis retracts according to the amount of retract (parameter No. 7741) and the speed (parameter No. 7740). As a result, a tool and a workpiece can be prevented from being damaged when the sudden alarm occurs. The axis whose amount of retract is set to “0” does not move. This improvement becomes effective to “General purpose retract” and “Retract of flexible synchronization control”. Moreover, the excitation of a servomotor is kept until the retract operation of the axis related to synchronization is completed, even when a servo alarm occurs in an axis not related to the synchronization in flexible synchronization control.

Conditions under the retract function with an alarm

The conditions under which the retract function with an alarm can be changed using the settings of bit 1 (ARE) of parameter No. 7703 and bit 2 (ARO) of parameter No. 7703.

The table below lists parameter settings and corresponding conditions.

ARE	ARO	Condition
1	0	During function mode
1	1	During function mode and automatic operation
0	0	During function mode or automatic operation
0	1	

Servo off

When the axis related to synchronization is servo off state while flexible synchronization control, an automatic operation is stopped. By setting bit 3 (FSV) of parameter No. 13421 to 1, an automatic operation can be stopped only when the axis related to synchronization moves.

Notes

1) Synchronization is not canceled with a reset.
2) Synchronization is executed even if the slave axis is in either of the following states:
   - Interlock
   - Feed hold
3) Synchronization is not maintained if the slave axis is in either of the following states:
   - Machine lock
   - Servo off
4) The master axis must not be a oscillation axis, an axis related to arbitrary angular axis control, an axis related to composite control, or an axis related to superimposed control.
   The slave axis must not be a oscillation axis, a PMC axis, an axis related to arbitrary angular axis control, an axis related to synchronization control, an axis related to composite control, or an axis related to superimposed control.
5) If an attempt is made to issue one of G28/G30/G53 during synchronization, alarm PS0010 is issued. To issue one of G28/G30/G53, cancel synchronization first. By setting parameter FRF to 1, they can be issued for a master axis.
6) Parameters for use with synchronization (Nos. 5660 to 5667, 5670 to 5677, 5680 to 5687, and 5690 to 5693) can be set in a part program that uses programmable parameter input (G10).
   Note that changes made to the parameters for a group that is already in synchronization mode do not take effect immediately. For the changes made to the parameters to take effect, it is necessary to turn off synchronization mode and turn it back on.
7) With a synchronization pulse, the position display for a slave axis is updated as follows:
   - Machine position display is updated.
     (It is updated with the amount of travel after acceleration/deceleration and, therefore, the display may appear not to be in synchronization.)
   - Absolute position display and relative position display are not updated.
     At the cancellation of synchronization, absolute position display and relative position display for a slave axis are updated by adding the amount of travel due to synchronization.
     If, however, bits 0 (ACA) to 3 (ACD) of parameter No. 5668 are set to 1 so that the machine coordinates for that group are not updated, the above-mentioned machine coordinate display, absolute coordinate display, and relative coordinate display are not updated.
8) If, during automatic operation, the synchronization mode is turned from off to on, alarm PS5242 "ILLEGAL AXIS NUMBER" is issued if a master axis number or a slave axis number is not set correctly.
9) If, during automatic operation, the synchronization mode is turned from off to on, alarm PS5243, "DATA OUTRANGE" is issued if a gear ratio is not set correctly.
10) If, during automatic operation, the synchronization mode is turned from off to on or from on to off, alarm PS5244, "TOO MANY DI ON" is issued if the mode signal is turned on or off after an M code is executed.
11) For a synchronization group for which a PMC axis is a master axis, be sure to set the controlled axis selection signal EAXn<Gn136> for PMC axis control to "1" before turning on the synchronization mode.
12) For a synchronization group for which spindle control with servo motor or spindle control with Cs contour control is a master axis, be sure to turn on the SV speed mode of spindle control with servo motor before turning on the synchronization mode.
13) For a synchronization group for which a PMC axis or spindle control with servo motor is a master axis, be sure to turn on the synchronization mode with an M code during automatic operation.
    If an attempt is made to turn on the synchronization mode not during automatic operation, alarm PS5245, "OTHERAXIS ARE COMMANDED" is issued.

14) For a synchronization group for which a PMC axis or spindle control with servo motor is a master axis, AI contour control I and AI contour control II are automatically turned off from the time the synchronization mode is turned on until it is turned off. However, when inter-path flexible synchronization mode select signal OVLN<Gn531.4> is assumed to be "1", advanced preview feed forward of a slave axis can be made effective. Advanced preview feed forward of a slave axis is controlled by a mode or modal of a slave axis.
15) For a synchronization group for which a PMC axis or spindle control with servo motor is a master axis, be sure to turn on the synchronization mode first. Also, for a synchronization group for which a PMC axis or spindle control with servo motor is a master axis, be sure to turn off the synchronization mode last.
16) If an attempt is made to turn on a synchronization group for which an PMC axis or spindle control with servo motor is a master axis when there exists a synchronization group for which a non-PMC or non-spindle control with servo motor, normal axis is a master axis, alarm PS5245 is issued.
17) In the synchronization mode or when the mode is turned on, alarm PS5245 is issued in the following cases:
    - The master and slave axes as synchronization axes overlap the EGB dummy axis.
    - The master and slave axes as synchronization axes overlap the oscillation axis.
    - The master and slave axes as synchronization axes overlap the axis related to arbitrary angular axis control.
    - The master and slave axes as synchronization axes overlap the axis related to composite control.
    - The master and slave axes as synchronization axes overlap the axis related to superimposed control.
    - The slave axis as a synchronization axis overlaps the axis related to synchronization control.
    - The reference position return mode is turned on (was turned on).
18) If an SV alarm is issued, the synchronization mode is automatically turned off after a deceleration stop.
19) The output pulses for the master axis in detection units are multiplied by the gear ratio, and the result is regarded the output pulses for the slave axis.
20) To synchronize with a spindle, using a servo motor, it is necessary to adjust the loop gain of the servo motor to the spindle and make sure that the position deviations are equal.
21) Actual cutting feedrate display does not take synchronization pulses into consideration.
22) Before the synchronization mode can be turned on or off, the master and slave axes must be stopped. If an attempt is made to turn on or off the synchronization mode when the axes are not stopped, alarm PS5244 is issued.
23) Specify retract only for the master axis. Specifying retract for the slave axis prevents normal retract operations. Set the retract operation parameter to 0 for the slave axis.
24) A retract operation for the master axis comes to halt if any of the following conditions occurs when retract is under flexible synchronization control. As for emergency stop, servo alarm, and alarm PS5245, the retraction operation comes to halt only when they occur within the path to which the synchronization axis belongs.
    - Emergency stop
    - Servo alarm
    - Overtravel alarm（referenced for individual axes）
    - Servo off（referenced for individual axes; The path to which the synchronization axis belongs is caused to stop.）
    - Alarm PS5245（OTHERAXIS ARE COMMANDED）

> ⚠ **WARNING**
>
> If two or more flexible synchronization control pairs are linked as stated below, the highest-order master axis (X1 axis) can be caused to stop by referencing the stop condition of the second slave axis (X3 axis) as viewed from the highest-order master axis (X1 axis). However, the stop condition of the third slave axis (X4) cannot be referenced.
>
> | | Master axis | | Slave axis |
> |---|---|---|---|
> | First pair | X1（Path 1） | → | X2（Path 2） |
> | Second pair | X2（Path 2） | → | X3（Path 3） |
> | Third pair | X3（Path 3） | → | X4（Path 4） |

25) In case parameter ART (No.7702#3)=1, the retract operation of “General purpose retract” / “Flexible synchronization control” has been changed from conventional one to new one in order to realize the retract operation at alarm generating. When an alarm occurs, a new retract operation is shown as follows.

Alarm	Conventional retract Bit 3 (ART) of parameter No. 7702=0	Retract with an alarm is enable. Bit 3 (ART) of parameter No. 7702=1
PW alarm	All retract operation stops.	Retract operation does not stop.
DS alarm / IE alarm (Malfunction prevent functions is enable.)	All retract operation stops because PW alarm occurs after this alarm.	Only the axis where alarm occurs stops. All retract operation stops for non-axis type alarm.
SV alarm	All retract operation stops.	Only the axis where alarm occurs stops. All retract operation stops for non-axis type alarm.

26) The retract function with an alarm does not perform a retract operation on the retract axis if an overtravel alarm or a servo alarm occurs on the retract axis. The retract function with an alarm does not perform a retract operation on the retract axis if an overtravel alarm or a servo alarm occurs on the slave axis in case of flexible synchronization control.
27) If a new alarm is issued during retraction with the retract function with an alarm, a retract operation is not performed.
28) After the end of retraction with a servo alarm, servo position control stops in 400 ms.
29) Please set increment system of the master axis and the slave axis to the same setting. It becomes operation different from the set ratio of the gear when assuming a different setting.
30) Flexible synchronization control cannot be used together with speed command in PMC axis control (Speed control).

Parameter

5660	**Master axis number (group A)**
5661	**Slave axis number (group A)**
5662	**Master axis number (group B)**
5663	**Slave axis number (group B)**
5664	**Master axis number (group C)**
5665	**Slave axis number (group C)**
5666	**Master axis number (group D)**
5667	**Slave axis number (group D)**

[Input type] Parameter input
[Data Input type] Word path
[Valid data range] 0 to Number of controlled axes or m × 100+n
(m: 1 to Number of paths, n: 1 to Number of controlled axes)
Specify both master and slave axis numbers.
[Example of setting]
1 to 24: Controlled axes on own path (for single-path systems only)
101 to 124: Controlled axes on path 1
201 to 224: Controlled axes on path 2

In inter-path flexible synchronization control, an axis of a path can be specified as the master axis of another path.

NOTE
In inter-path flexible synchronization control, an axis of any path cannot be specified as the slave axis of another path.

	#7	#6	#5	#4	#3	#2	#1	#0
5668					ACD	ACC	ACB	ACA

[Input type] Parameter input
[Data Input type] Bit axis

#0 ACA Update of the machine coordinates of flexible synchronization control group A is:
0 : Executed.
1 : Not executed.

#1 ACB Update of the machine coordinates of flexible synchronization control group B is:
0 : Executed.
1 : Not executed.

#2 ACC Update of the machine coordinates of flexible synchronization control group C is:
0 : Executed.
1 : Not executed.

#3 ACD Update of the machine coordinates of flexible synchronization control group D is:
0 : Executed.
1 : Not executed.

NOTE
The machine coordinates update is not done though the slave axis operates on the motor.
In this case, if an automatic reference position return to origin is done after the synchronization mode is canceled, the alarm of DS0405, "ZERO RETURN END NOT ON REF" is issued.
Please use a manual reference position return to origin to do the return to origin.

5670	M code number for turning on the flexible synchronization control mode(group A)
5671	M code number for turning off the flexible synchronization control mode(group A)
5672	M code number for turning on the flexible synchronization control mode(group B)

5673	M code number for turning off the flexible synchronization control mode(group B)
5674	M code number for turning on the flexible synchronization control mode(group C)
5675	M code number for turning off the flexible synchronization control mode(group C)
5676	M code number for turning on the flexible synchronization control mode(group D)
5677	M code number for turning off the flexible synchronization control mode(group D)

[Input type] Parameter input
[Data Input type] Word path
[Valid data range] 1 to 999

Specify an M code for turning on or off the flexible synchronization control mode for an automatic operation.

5680	Numerator determining gear ratio for flexible synchronization(group A)
5681	Denominator determining gear ratio for flexible synchronization(group A)
5682	Numerator determining gear ratio for flexible synchronization(group B)
5683	Denominator determining gear ratio for flexible synchronization(group B)
5684	Numerator determining gear ratio for flexible synchronization(group C)
5685	Denominator determining gear ratio for flexible synchronization(group C)
5686	Numerator determining gear ratio for flexible synchronization(group D)
5687	Denominator determining gear ratio for flexible synchronization(group D)

[Input type] Parameter input
[Data Input type] 2 word path
[Valid data range] -99999999 to 99999999

Specify a gear ratio between the master and slave axes.

5690	Index to gear ratio denominator for flexible synchronization(group A)
5691	Index to gear ratio denominator for flexible synchronization(group B)
5692	Index to gear ratio denominator for flexible synchronization(group C)
5693	Index to gear ratio denominator for flexible synchronization(group D)

[Input type] Parameter input
[Data Input type] Byte path
[Valid data range] 0 to 8

Specify an index to the denominator of a gear ratio between the master and slave axes.
Let p, q, and k be, respectively, a denominator determining gear ratio for flexible synchronization, numerator determining gear ratio for flexible synchronization, and index to the gear ratio denominator for flexible synchronization:

The gear ratio is $\dfrac{q}{p \times 10^k}$

	#7	#6	#5	#4	#3	#2	#1	#0
7702					ART			

[Input type] Parameter input

[Data type] Bit path

#3 ART The retract function executed when an alarm is issued is:
0: Disabled.
1: Enabled.
When an alarm is issued, a retract operation is performed with a set feedrate and travel distance (parameters Nos. 7740 and 7741).

> **NOTE**
> If a servo alarm is issued for other than the axis along which a retract operation is performed, the servo activating current is maintained until the retract operation is completed.

	#7	#6	#5	#4	#3	#2	#1	#0
7703						ARO	ARE	

[Input type] Parameter input
[Data type] Bit path

#1 ARE In the retract function by an alarm, the tool retracts:
0: During the function mode (refer to NOTE.2) or automatic operation (automatic operation signal OP <Fn000.7> = “1”).
1: During the function mode (refer to NOTE.2).

#2 ARO In the retract function by an alarm, the tool retracts :
0: During the function mode (refer to NOTE.2).
1: During the function mode (refer to NOTE.2) and automatic operation (automatic operation signal OP = “1”).

> **NOTE**
> This parameter is effective when bit 1 (ARE) of parameter No. 7703 is set to 1.

The following table lists the parameter settings and corresponding operation.

ARE	ARO	Operation
1	0	During function mode
1	1	During function mode and automatic operation
0	0	During function mode or automatic operation
0	1	

> **NOTE**
> 1 Parameters ARE and ARO are valid when bit 3 (ART) of parameter No. 7702 is set to 1 (when the retract function executed when an alarm is issued).
> 2 The function mode described in the parameter ARE and ARO shows that the following functions are activated.
> - Electronic gear box(EGB)
> - Flexible synchronization control

	#7	#6	#5	#4	#3	#2	#1	#0
13421				FPA	FSV	FRF		

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> Set these parameters for the first path only. It will be effective to all paths.

#2 FRF If G27/G28/G29/G30/G53 is specified during flexible synchronization control, alarm PS0010 is:
0: Issued.
1: Is not issued. Commands to the master axis are possible.
Even if, however, parameter bit FRF is set to 1, and G28 is specified for the master axis in the state in which the reference position of the master axis subject to flexible synchronization control is not established, or if G27/G28/G29/G30/G53 is specified for the slave axis, alarm PS5381 is issued.

#3 FSV When the axis related to synchronization is servo off state while flexible synchronization control or inter-path flexible synchronization control, an automatic operation is:
0: Stopped.
1: Stopped if the axis related to synchronization moves.

> **NOTE**
> In inter-path flexible synchronization control, this parameter becomes effective when parameter FCN (No.13421#1) is set to 1.

#4 FPA For a synchronization group for which a PMC axis is a master axis, when the controlled axis selection signal EAXn<Gn136> for PMC axis control is set to "1" after starting flexible synchronization control mode, and the master axis is specified by PMC axis control, or for a synchronization group for which spindle control with servo motor is a master axis, when SV speed control mode of the master axis is turned on after starting flexible synchronization control mode:
0: Alarm is not generated.
1: Alarm PS5381 ”INVALID COMMAND IN FSC MODE” is generated.

This function generates the alarm when flexible synchronization control is operated by in the incorrect procedure.

> ⚠**CAUTION**
> When this parameter is set to 0, it operates without generating the alarm even if flexible synchronization control mode is started according to a procedure not correct. However, the position shift of a slave axis may occur.

Signal

Flexible synchronization control mode selection signals MTA,MTB,MTC,MTD <Gn197.0 to Gn197.3>

[Classification] Input signal
[Function] These signals select flexible synchronization control.
[Operation] 1) Synchronization is started by setting one of these signals to "1".
2) Synchronization is canceled by setting one of these signals to "0".
MTA: Synchronization of group A is selected.
MTB: Synchronization of group B is selected.

MTC: Synchronization of group C is selected.
MTD: Synchronization of group D is selected.

Flexible synchronization control mode status signals
MFSYNA, MFSYNB, MFSYNC, MFSYND <Fn197.0 to Fn197.3>

[Classification] Output signal

[Function] These signals are used to check that the groups selected with the flexible synchronization control mode selection signals are actually switched to that mode.

[Operation]
1) When the synchronization mode actually becomes effective to a group, the corresponding one of these signals is set to "1".
2) When the synchronization mode is actually canceled for a group, the corresponding one of these signals is set to "0".

MFSYNA: Group A is in the synchronization mode.
MFSYNB: Group B is in the synchronization mode.
MFSYNC: Group C is in the synchronization mode.
MFSYND: Group D is in the synchronization mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn197					MTD	MTC	MTB	MTA

	#7	#6	#5	#4	#3	#2	#1	#0
Fn197					MFSYND	MFSYNC	MFSYNB	MFSYNA

Alarm and message

Number	Message	Description
PS0010	IMPROPER G-CODE	In case parameter FRF (No.13421#2)=1, One of G27/G28/G29/G30/G30/G53 is commanded to a master axis or a slave axis in flexible synchronization control mode.
PS5242	ILLEGAL AXIS NUMBER	A master axis number or a slave axis number was not set correctly when the flexible synchronization control mode was turned from off to on during automatic operation.
PS5243	DATA OUTRANGE	A gear ratio was not set correctly when the flexible synchronization control mode was turned from off to on during automatic operation.
PS5244	TOO MANY DI ON	- During automatic operation, the mode signal was not turned on or off if the flexible synchronization control mode was turned from off to on or from on to off, after an M code was executed. - An attempt was made to turn on or off the flexible synchronization control mode when the axis was not stopped.

Number	Message	Description
PS5245	OTHERAXIS ARE COMMANDED	- For a flexible synchronization control group for which a PMC axis or spindle control with servo motor was a master axis, an attempt was made to turn on the synchronization mode during time other than automatic operation. - An attempt was made to turn on a synchronization group for which an PMC axis or spindle control with servo motor was a master axis when there existed a flexible synchronization control group for which a non-PMC or non-spindle control with servo motor, normal axis was a master axis. - The master and slave axes as synchronization axes overlap the EGB dummy axis. - The master and slave axes as synchronization axes overlap the oscillation axis. - The master and slave axes as synchronization axes overlap the axis related to arbitrary angular axis control. - The master and slave axes as synchronization axes overlap the axis related to composite control. - The master and slave axes as synchronization axes overlap the axis related to superimposed control. - The slave axis as a synchronization axis overlaps the axis related to synchronization control. - The reference position return mode is turned on (was turned on).
PS5381	INVALID COMMAND IN FSC MODE	An attempt was made to issue the following commands: - When the reference position for the master axis under flexible synchronization control has not been established, G28 command for the master axis. - G27/G28/G29/G30/G53 command for a slave axis.
DS405	ZERO RETURN END NOT ON REF	In case parameter ACA to ACD (No.5668#0 to #3)=1, a slave axis is executed an automatic reference position return after canceling synchronization mode. Please use a manual reference position return to origin to do the return to origin.

1.12.2 Automatic Phase Synchronization for flexible synchronization Control

Overview

This function applies acceleration/deceleration when the start or cancellation of synchronization is specified in flexible synchronization control.

This acceleration/deceleration allows synchronization to be started or canceled while the tool is moving along the master axis.

This function can also execute automatic phase synchronization so that the slave axis machine coordinate position at the start of synchronization matches the machine coordinate system zero point of the master axis (the machine coordinate is 0).

Acceleration/deceleration

When bit 0 (PHA) to 3 (PHD) of parameter No. 5669 is set to 1, acceleration/deceleration is applied when the start or cancellation of synchronization is specified.

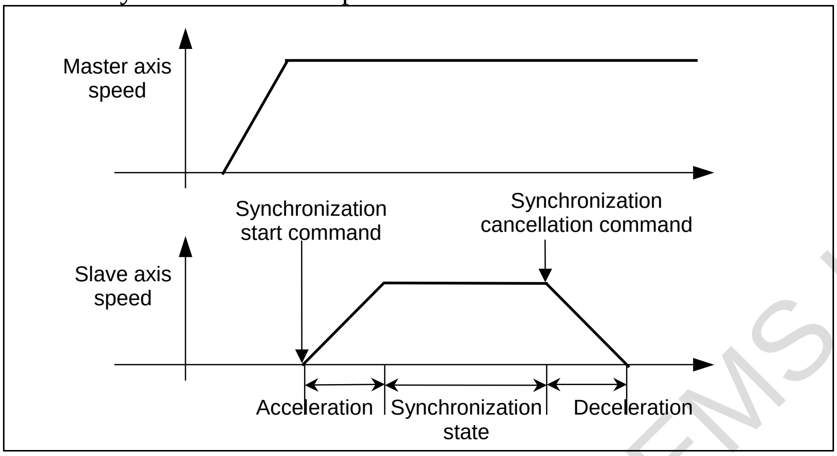


Fig. 1.12.2 (a)

Synchronization start
Command sequence

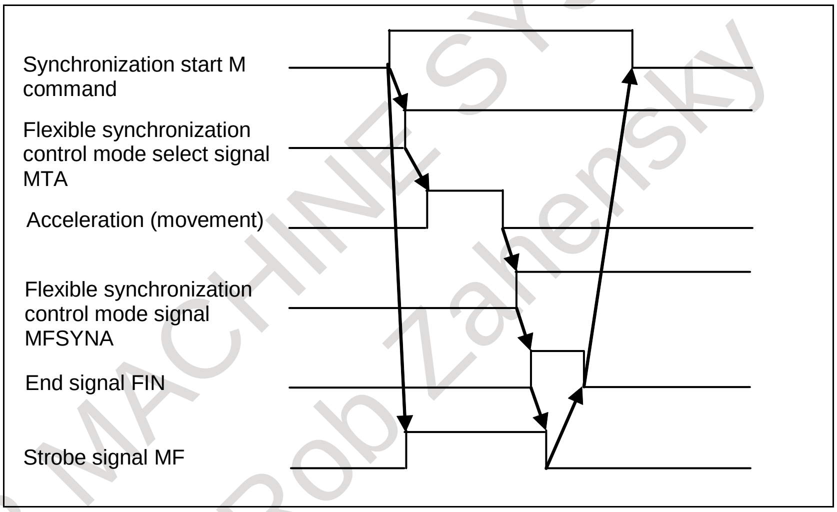


Fig. 1.12.2 (b)

1. When the M code for turning the synchronization control mode on is specified, and the flexible synchronization control mode selection signal MTA, MTB, MTC, or MTD which corresponds to the M code is set to "1", synchronization starts.
2. The tool moves along the slave axis at the acceleration rate set in parameters Nos. 1420 and 13425 to 13428. Once the synchronization feedrate is reached, the flexible synchronization control mode status signal MFSYNA, MFSYNB, MFSYNC, or MFSYND becomes "1".
3. When the flexible synchronization control mode status signal becomes "1", the completion of the M code for starting synchronization is returned.

Synchronization cancellation
Command sequence

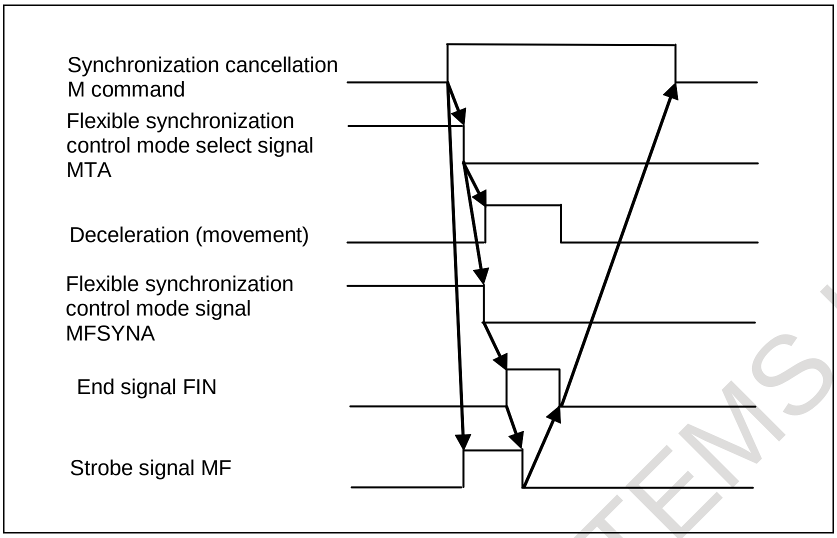


Fig. 1.12.2 (c)

1. Move the tool away from the workpiece before cancellation.
2. When the M code for turning the flexible synchronization control mode off (canceling synchronization) is specified, and the flexible synchronization control mode selection signal which corresponds to the M code is set to "0", synchronization cancellation starts. The tool decelerates along the slave axis at the acceleration rate set in parameters Nos. 1420 and 13425 to 13428.
3. When deceleration starts, the flexible synchronization control mode status signal becomes "0".
4. The completion of the M code for canceling synchronization is returned.

NOTE
1 Linear acceleration/deceleration is applied to synchronization start/cancellation.
2 When time constant parameters No. 13425 to 13428 are 0, acceleration/deceleration is not applied.
3 The next block is not executed until deceleration is completed during automatic operation.

Automatic phase synchronization

When bit 0 (PHA) to 3 (PHD) of parameter No. 5669 is set to 1, and the flexible synchronization control automatic phase synchronization signal AUTPHA, AUTPHB, AUTPHC, or AUTPHD is set to "1", automatic phase synchronization is executed after acceleration/deceleration applied at the start of synchronization is completed. Phase synchronization is automatically executed so that the slave axis machine coordinate position at the start of synchronization (when the flexible synchronization control mode selection signal is set to "1") matches the machine coordinate system zero point of the master axis (the machine coordinate is 0).
When automatic phase synchronization for flexible synchronization control is enabled, acceleration/ deceleration is applied when the start or cancellation of synchronization is specified.

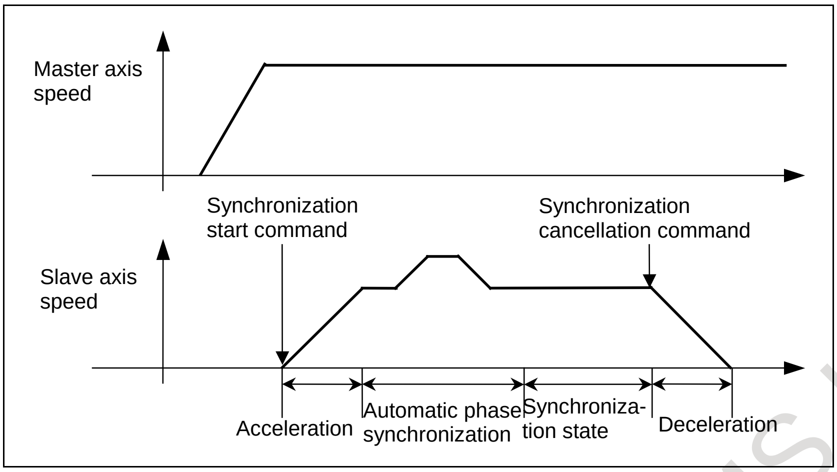


Fig. 1.12.2 (d)

Command sequence

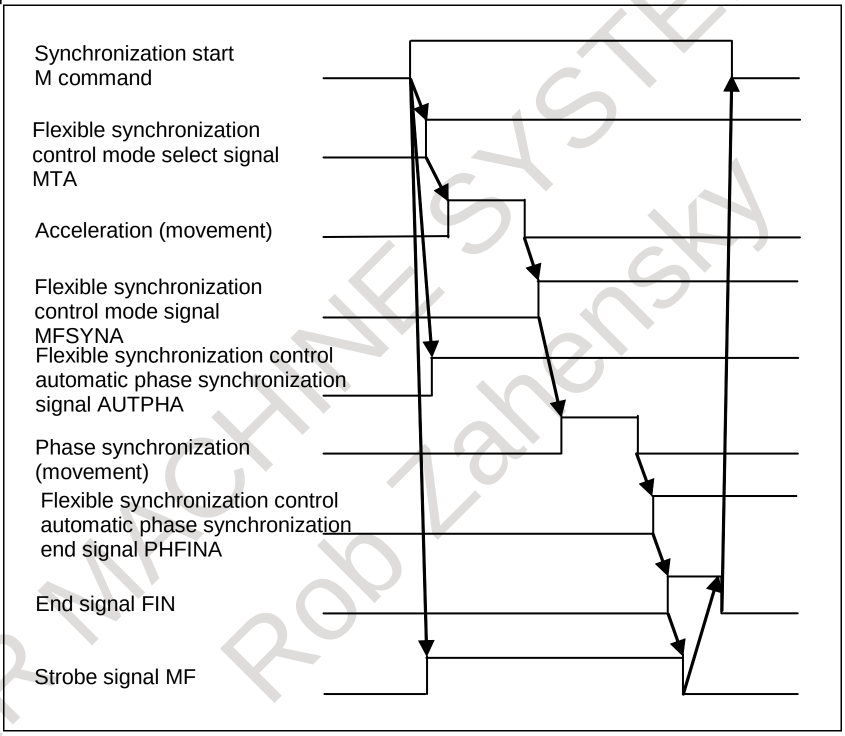


Fig. 1.12.2 (e)

1. When the M code for turning the flexible synchronization control mode on is specified, and the flexible synchronization control mode selection signal which corresponds to the M code is set to "1", synchronization starts.
   The flexible synchronization control automatic phase synchronization signal is also set to "1".
2. The tool moves along the slave axis at the acceleration rate set in parameters Nos. 1420 and 13425 to 13428.
3. Once the synchronization feedrate is reached, the flexible synchronization control mode status signal becomes "1".
4. When the flexible synchronization control mode status signal becomes "1", phase synchronization is executed automatically.
5. When phase synchronization is completed, the flexible synchronization control phase synchronization end signal PHFINA, PHFINB, PHFINC, or PHFIND becomes "1".
6. When the flexible synchronization control phase synchronization end signal becomes "1", the completion of the M code for starting synchronization is returned.

> **NOTE**
> 1 Set the feedrate in automatic phase synchronization in parameters Nos. 13429 to 13432.
> 2 Specify the movement direction for automatic phase synchronization in bit 0 (DIA) to 3 (DID) of parameter No. 13420. (Rotation axis only)
> 3 Linear acceleration/deceleration is applied to automatic phase synchronization. (The acceleration rate is the same as that at the synchronization start or cancellation.)
> 4 The feedrate along the slave axis is obtained by superposing the feedrate in automatic phase synchronization onto the feedrate in synchronization with the movement along the master axis. In consideration of this superposition, set a position deviation limit in parameter No. 1828.
> 5 By setting parameters Nos. 13433 to 13436, the position at which the phase of the slave axis is matched can be shifted from the machine coordinate system zero point of the master axis.
> 6 If the automatic phase synchronization signal is set to "0", then "1" again during synchronization, automatic phase synchronization is executed again.
> 7 When the flexible synchronization control automatic phase synchronization signal is set to "1" during synchronization (when the flexible synchronization control mode status signal is "1"), automatic phase synchronization is also executed if the tool does not move along the master axis. The tool moves only along the slave axis.

Change of a gear ratio

A gear ratio can be changed during automatic operation by changing values of parameters for the gear ratio (parameters Nos.5680 to 5687 and 5690 to 5693) using programmable parameter input (G10).

> **NOTE**
> 1 When the values of the parameters for a gear ratio are changed, the flexible synchronization control mode status signal becomes "0", and acceleration /deceleration is applied. When the synchronization feedrate is reached, the signal becomes "1" again.
> 2 If automatic phase synchronization is executed after a gear ratio is changed, the position for automatic phase synchronization is as follows:
> Master axis: Machine coordinate system zero point
> (value set in parameters Nos. 13433 to 13436)
> Slave axis: Position when synchronization starts
> (when the flexible synchronization mode selection signal is set to "1")
> 3 If you want to change the values of the parameters for two or more gear ratios, stop the machine before changing them.

Speed-up of automatic phase synchronization

During accelerating of a slave axis that begins synchronization, automatic phase synchronization or decelerating of the slave axis that cancels synchronization, it is possible to command other program. This function is enabled to set the bit 1 (PHS) of parameter No.5694 to 1.

(1) During accelerating of a slave axis that begins synchronization

In command of M code for beginning synchronization, when flexible synchronization control mode selecting signals FSYSA, FSYSB, FSYSC, FSYSD <Fn553.4, Fn553.5, Fn553.6, Fn553.7> is ‘1’, it is possible to return end signal FIN <Gn004.3>. It is possible to command programs though a slave axis is accelerating until flexible synchronization control mode status signals MFSYNA, MFSYNB, MFSYNC, MFSYND <Fn197.0, Fn197.1, Fn197.2, Fn197.3> are "1" after being set flexible synchronization control mode selecting signal to "1".

(2) During automatic phase synchronization

When flexible synchronization control automatic phase synchronization signals AUTPHA, AUTPHB, AUTPHC, AUTPHD <Gn381.0 to Gn381.3> are set to "1" by M code In automatic operation, it is possible to return end signal FIN <Gn004.3> at once. When flexible synchronization control automatic phase synchronization signals are set to "1" by M code for turning on synchronization, return end signal after being set flexible synchronization control mode selecting signal to "1". It is possible to command programs though a slave axis is accelerating or decelerating until flexible synchronization control phase synchronization end signals PHFINA, PHFINB, PHFINC, PHFIND <Fn381.0 to Fn381.3> is "1" after being set flexible synchronization control automatic phase synchronization signal to "1". In previous in the block that would like to be executed with a master axis suitable for the phase of a slave axis, confirm flexible synchronization control phase synchronization end signal.

(3) During decelerating of a slave axis that cancels synchronization

Meanwhile, other program instructions can be executed by decelerating a slave axis for canceling synchronization before instruction M code of canceling synchronization (Parameters No.5671, No.5673, No.5675 and No.5677). Parameter No.5696 is set decelerating start M code for turning off synchronization. When M code for beginning deceleration for turning off synchronization is specified, the deceleration of the slave axis is begun. The block of M code for beginning deceleration for turning off synchronization finishes at once, and advances to next block. It is possible to command programs though a slave axis is decelerating until executing M code for turning off synchronization after executing deceleration start M code for turning off synchronization. M code for beginning deceleration for turning off synchronization can be executed in two or more M codes in one block.

Hob command by flexible synchronization control

When G81 or G81.4 for turning on synchronization is specified, the block of instruction is finished at once, and advances to next block. It is possible to command programs though a slave axis is accelerating to synchronous speed until flexible synchronization control mode status signal is "1". During decelerating of a slave axis that cancels synchronization and automatic phase synchronization are similar to usual flexible synchronization control.

NOTE

1. When a master axis or slave axis would like to specify in finished synchronization, confirm flexible synchronization control mode status signal.
2. When a master axis or slave axis would like to specify in finished phase synchronization, confirm flexible synchronization control phase synchronization end signal.
3. Decelerating start M code for turning off synchronization (Parameter No.5696) does not output code signal and strobe signal.
4. When decelerating start M code for turning off synchronization is specified, synchronization cannot begin again if synchronization can begin again only after it is cancelled first. When a gear ratio is changed after specifying decelerating start M code for turning off synchronization, alarm PS5381 "INVALID COMMAND IN FSC MODE" is issued.

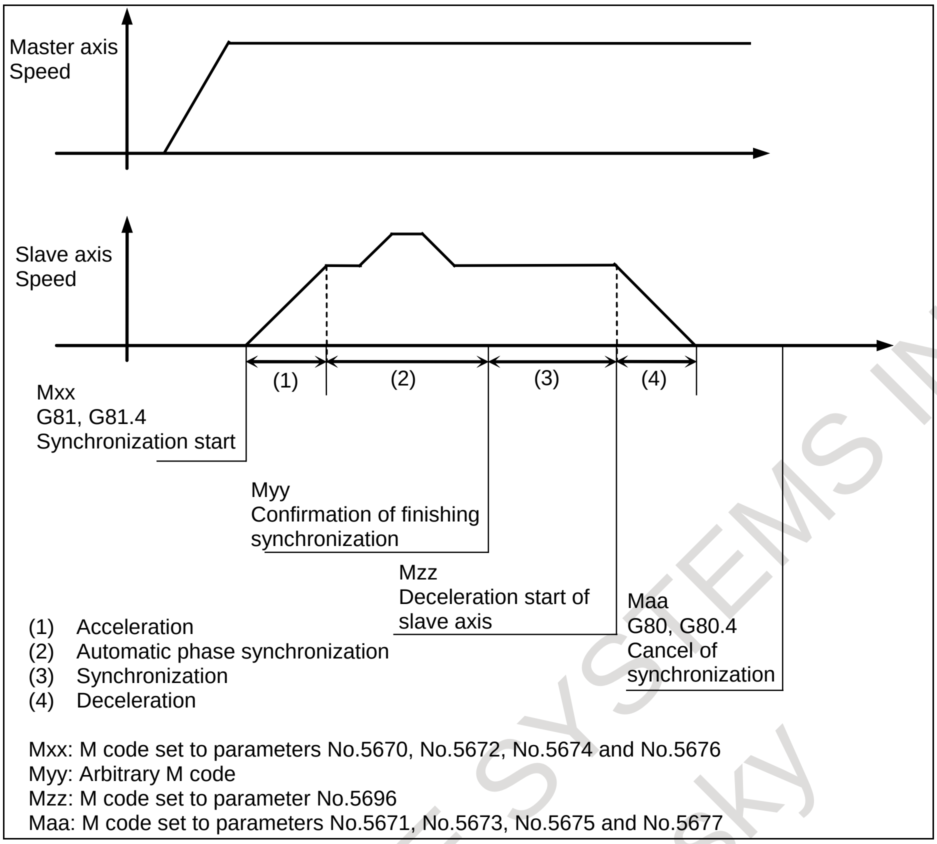
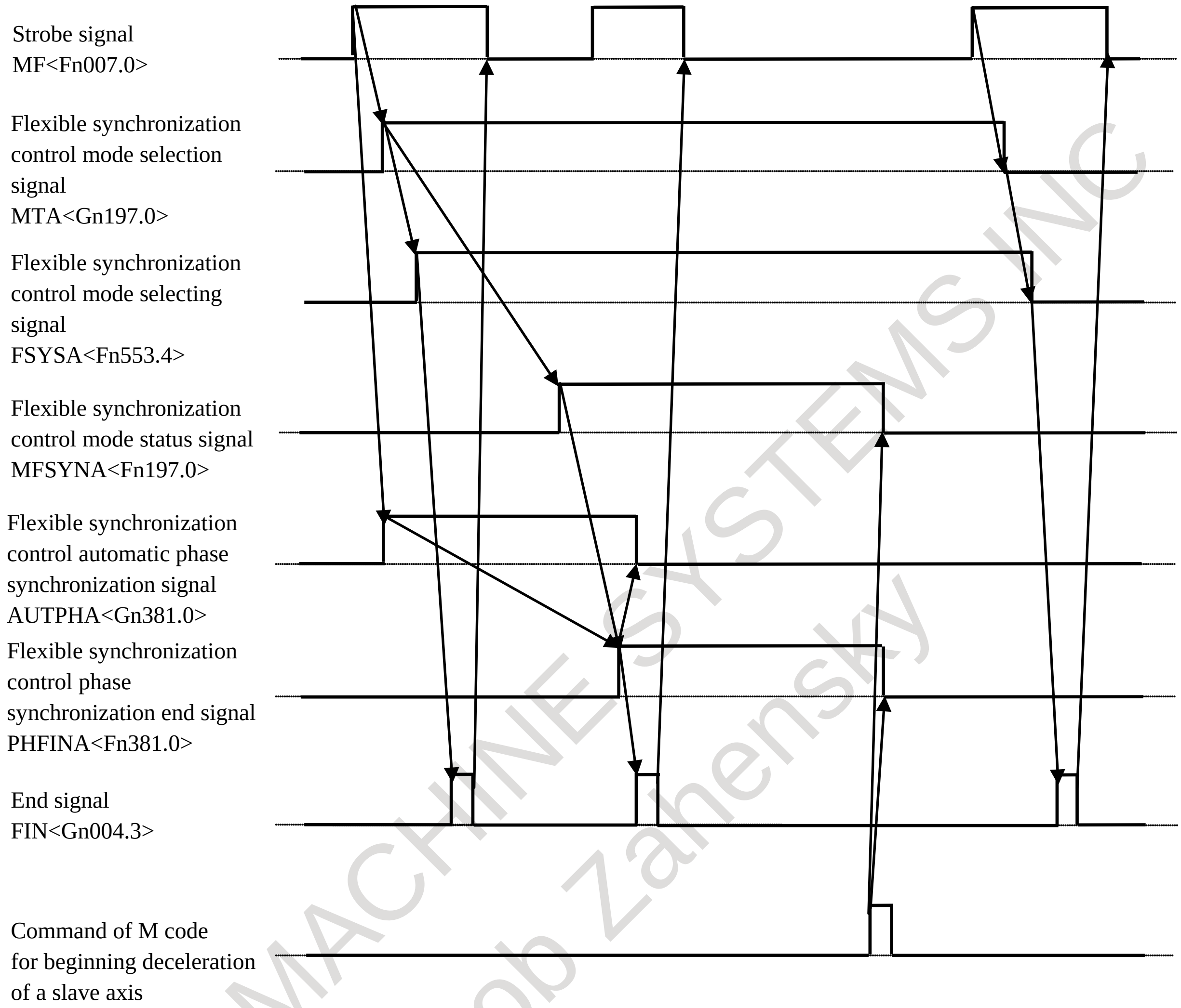
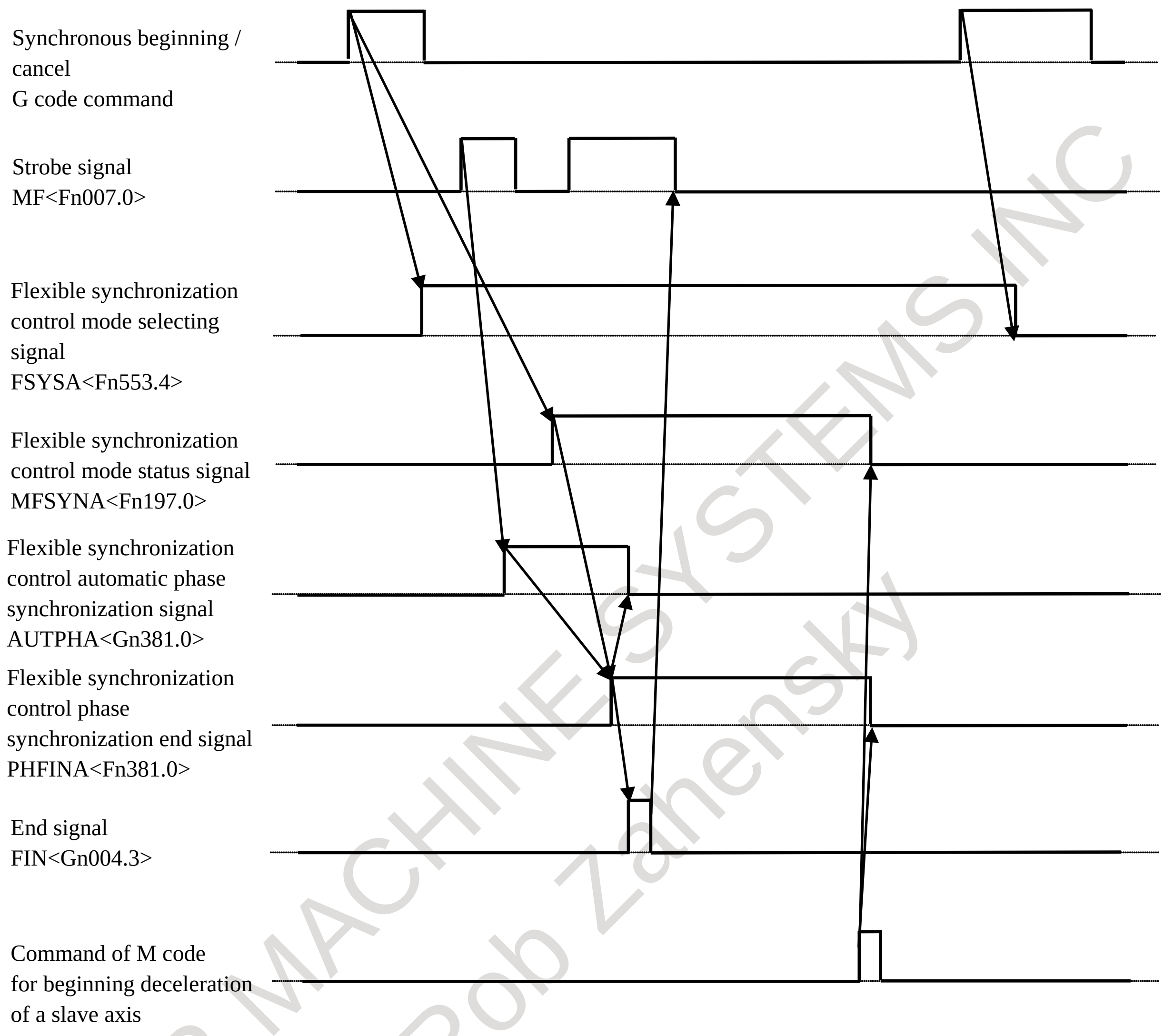


Fig. 1.12.2 (f) Type of acceleration / deceleration and automatic phase synchronization

Example of time chart high speed type for acceleration / deceleration and automatic phase synchronization



Example of time chart high speed type for acceleration / deceleration and automatic phase synchronization (Hob command)



Signal

Flexible synchronization control mode selection signals MTA, MTB, MTC, MTD <Gn197.0 to Gn197.3>

[Classification] Input signal

[Function] These signals select flexible synchronization control.

[Operation]
1) Synchronization starts when these signals become "1".
2) Synchronization is canceled when these signals become "0".

MTA: Selects synchronization for group A.
MTB: Selects synchronization for group B.
MTC: Selects synchronization for group C.
MTD: Selects synchronization for group D.

Flexible synchronization control mode status signals MFSYNA, MFSYNB, MFSYNC, MFSYND <Fn197.0 to Fn197.3>

[Classification] Output signal

[Function] These signals check whether the group selected by a flexible synchronization control mode selection signal has actually entered the mode.

[Operation] These signals become "1" when:

- Acceleration/deceleration is applied at the start of synchronization and the synchronization feedrate is reached.

These signals become "0" when:

- Deceleration starts at synchronization cancellation.
- The gear ratio is changed during automatic operation.

MFSYNA: Synchronization select switching for group A has been accepted.
MFSYNB: Synchronization select switching for group B has been accepted.
MFSYNC: Synchronization select switching for group C has been accepted.
MFSYND: Synchronization select switching for group D has been accepted.

Flexible synchronization control automatic phase synchronization signals
AUTPHA, AUTPHB, AUTPHC, AUTPHD <Gn381.0 to Gn381.3>

[Classification] Input signal

[Function] These signals select automatic phase synchronization for flexible synchronization control.

[Operation] When these signals are "0", automatic phase synchronization is not executed.
When these signals are "1", automatic phase synchronization is executed after acceleration/deceleration applied at the start of synchronization start is completed.
(When the flexible synchronization control mode status signal is "1", automatic phase synchronization is executed.)
When these signals are set to 0, then "1" again during synchronization, automatic phase synchronization is executed again.

AUTPHA: Selects automatic phase synchronization for group A.
AUTPHB: Selects automatic phase synchronization for group B.
AUTPHC: Selects automatic phase synchronization for group C.
AUTPHD: Selects automatic phase synchronization for group D.

Flexible synchronization control phase synchronization end signals
PHFINA, PHFINB, PHFINC, PHFIND <Fn381.0 to Fn381.3>

[Classification] Output signal

[Function] These signals notify that automatic phase synchronization for flexible synchronization control is completed.

[Operation] These signals become "1" when:

- Automatic phase synchronization is completed.

These signals become "0" when:

- The flexible synchronization control mode status signal becomes "0".

PHFINA: Notifies that automatic phase synchronization for group A is completed.
PHFINB: Notifies that automatic phase synchronization for group B is completed.
PHFINC: Notifies that automatic phase synchronization for group C is completed.
PHFIND: Notifies that automatic phase synchronization for group D is completed.

Flexible synchronization control mode selecting signals
FSYSA, FSYSB, FSYSC, FSYSD <Fn553.4 to Fn553.7>

[Classification] Output signal

[Function] These signals notify that the change to flexible synchronization control mode is begun.

[Operation] These signals become "1" when:

- The change to flexible synchronization control mode is begun.

These signals become "0" when:

- Flexible synchronization control mode is canceled

> **NOTE**
> If the bit 1 (PHS) of parameter No.5694 is set to 1, this signal is enabled.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn197					MTD	MTC	MTB	MTA
Gn381					AUTPHD	AUTPHC	AUTPHB	AUTPHA
Fn197					MFSYND	MFSYNC	MFSYNB	MFSYNA
Fn381					PHFIND	PHFINC	PHFINB	PHFINA
Fn553	FSYSD	FSYSC	FSYSB	FSYSA				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5669					PHD	PHC	PHB	PHA

[Input type] Parameter input
[Data Input type] Bit path

#0 PHA The automatic phase synchronization for flexible synchronization control of group A is:
0: Disabled.
1: Enabled.

#1 PHB The automatic phase synchronization for flexible synchronization control of group B is:
0: Disabled.
1: Enabled.

#2 PHC The automatic phase synchronization for flexible synchronization control of group C is:
0: Disabled.
1: Enabled.

#3 PHD The automatic phase synchronization for flexible synchronization control of group D is:
0: Disabled.
1: Enabled.

> **NOTE**
> When this parameter is set, acceleration/deceleration upon a synchronization start or synchronization cancellation is enabled.
> For automatic positioning, set the automatic phase synchronization signal for each group to "1".

	#7	#6	#5	#4	#3	#2	#1	#0
5694							PHS	

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 PHS Speed-up of automatic phase synchronization for flexible synchronization control is:
0: Disabled.
1: Enabled.

5696	M code for beginning deceleration for canceling synchronization

[Input type] Parameter input
[Data type] 2 Word path
[Valid data range] 0 to 99999999

This parameter set M code for beginning deceleration for canceling synchronization.
The M code of group A : Number set to this parameter
The M code of group B : Number set to this parameter + 1
The M code of group C : Number set to this parameter + 2
The M code of group D : Number set to this parameter + 3
When this parameter is set to 0, the M code is disabled.

> **NOTE**
> 1 If the bit 1 (PHS) of parameter No.5694 is set to 1, this parameter is enabled.
> 2 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

	#7	#6	#5	#4	#3	#2	#1	#0
13420					DID	DIC	DIB	DIA

[Input type] Parameter input
[Data type] Bit path

#0 DIA The movement direction of the automatic phase synchronization of group A is:
0: + direction.
1: - direction.

#1 DIB The movement direction of the automatic phase synchronization of group B is:
0: + direction.
1: - direction.

#2 DIC The movement direction of the automatic phase synchronization of group C is:
0: + direction.
1: - direction.

#3 DID The movement direction of the automatic phase synchronization of group D is:
0: + direction.
1: - direction.

13425	Acceleration/deceleration time constant of the slave axis when synchronization is started/canceled (group A)

13426	Acceleration/deceleration time constant of the slave axis when synchronization is started/canceled (group B)

13427	Acceleration/deceleration time constant of the slave axis when synchronization is started/canceled (group C)

13428	Acceleration/deceleration time constant of the slave axis when synchronization is started/canceled (group D)

[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 4000

These parameters set the acceleration/deceleration time constants of the slave axis subject to automatic phase synchronization for flexible synchronization control.
The acceleration when synchronization is started/canceled will be as follows:
Acceleration = parameter No.1420 / parameters Nos.13425 to 13428

13429	Automatic phase synchronization feedrate for the slave axis (group A)

13430	Automatic phase synchronization feedrate for the slave axis (group B)

13431	Automatic phase synchronization feedrate for the slave axis (group C)

13432	Automatic phase synchronization feedrate for the slave axis (group D)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

These parameters set the automatic phase synchronization feedrates for the slave axis subject to automatic phase synchronization.
These feedrates are superimposed on the feedrate synchronized to the master axis.
If the setting of one of the parameters is 0, the automatic phase synchronization feedrate for the corresponding group will be 6 (mm/min).

13433	Machine coordinates of the master axis used as the reference for phase synchronization (group A)

13434	Machine coordinates of the master axis used as the reference for phase synchronization (group B)

13435	Machine coordinates of the master axis used as the reference for phase synchronization (group C)

13436	Machine coordinates of the master axis used as the reference for phase synchronization (group D)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

These parameters set the machine coordinates of the master axis used as the reference for phase synchronization. If the setting of this parameter is 0, the origin position (coordinates: 0) of the machine coordinate system of the master axis will be the reference position for automatic phase synchronization.

Alarm and message

Number	Message	Description
PS5242	ILLEGAL AXIS NUMBER	- A master axis number or a slave axis number was not set correctly when the flexible synchronization control mode was turned from off to on during automatic operation. - Automatic phase synchronization is effective, and a set increment system of the master axis and the slave axis is different. - Automatic phase synchronization is effective, and the setting of the rotation axis of the master axis or the slave axis is illegal.
PS5381	INVALID COMMAND IN FSC MODE	- The gear ratio is changed without canceling synchronization after specifying M code of beginning deceleration for canceling synchronization.

Notes

NOTE

1 The next block is not executed until acceleration/deceleration at the start or cancellation of synchronization is completed during automatic operation.
2 Due to an error produced when the output pulses for the slave axis are calculated, the phase of the slave axis may not be matched by least input increment. This error is not accumulated.
3 If a retract operation is performed during automatic phase synchronization, the automatic phase synchronization is stopped and the retract operation is performed. After the completion of the retract operation, restarting the program resumes the automatic phase synchronization. However, no request for automatic phase synchronization is accepted during a retract operation.
4 It is necessary to set increment system of the master axis and the slave axis to the same setting. If flexible synchronization is started when it is different setting, alarm PS5242 "OTHER AXIS ARE COMMANDED" is issued.
5 When the master axis and the slave axis are the following settings, automatic phase match cannot be used. If flexible synchronization is started, alarm PS5242 "OTHER AXIS ARE COMMANDED" is issued.
   - Either the master axis or the slave axis is rotation axis, and the type of rotation axis is A type.
   - Both the master axes and the slave axes are the rotation axes, and the type of the rotation axis is different.
   - Both the master axes and the slave axes are the rotation axes of type A, and the value of parameter No.1260 is different.

1.12.3 Variable Acceleration Function in Automatic Phase Synchronization for Flexible Synchronization Control

Outline

In automatic phase synchronization for flexible synchronization control, acceleration of slave axis can be changed depending on the speed in during synchronization start/cancellation.

Explanation

Over the base speed, output characteristic of a spindle motor is constant and its torque characteristic decreases in inverse proportion to the speed. Therefore if the target speed is over the base speed in the linear acceleration pattern, low acceleration must be set depending on torque characteristic at the target speed, and motor torque can not be used efficiently in the low speed area. If this variable acceleration function is used, the acceleration is constant (specified by parameter No.7778) in lower speed area than parameter No.5697 and the acceleration decreases in inverse proportion to the speed in higher speed area than parameter No.5697. To use this function, set the parameter PAV(No.5694#2) to 1.

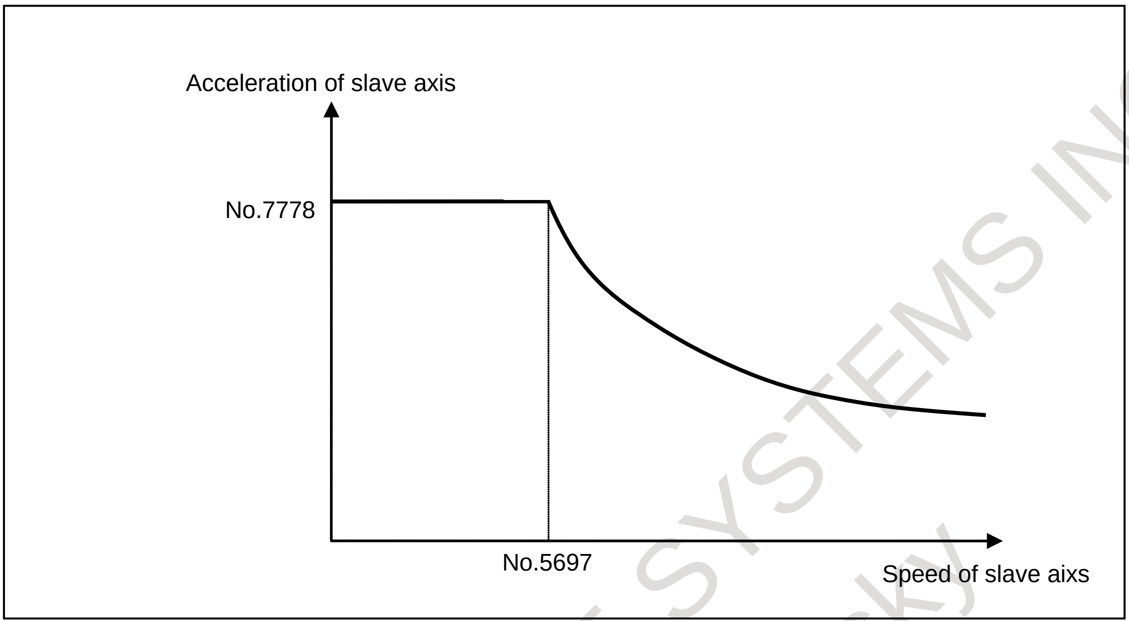


Fig. 1.12.3 (a) Acceleration of slave in synchronization start/cancellation

Speed of slave axis

Speed of slave axis used for acceleration calculation of this function is the number of rotation calculated by the shift amount per one rotation of a rotary axis specified by the parameter No.1260. Set the value to parameter No.5697 considering this calculation.

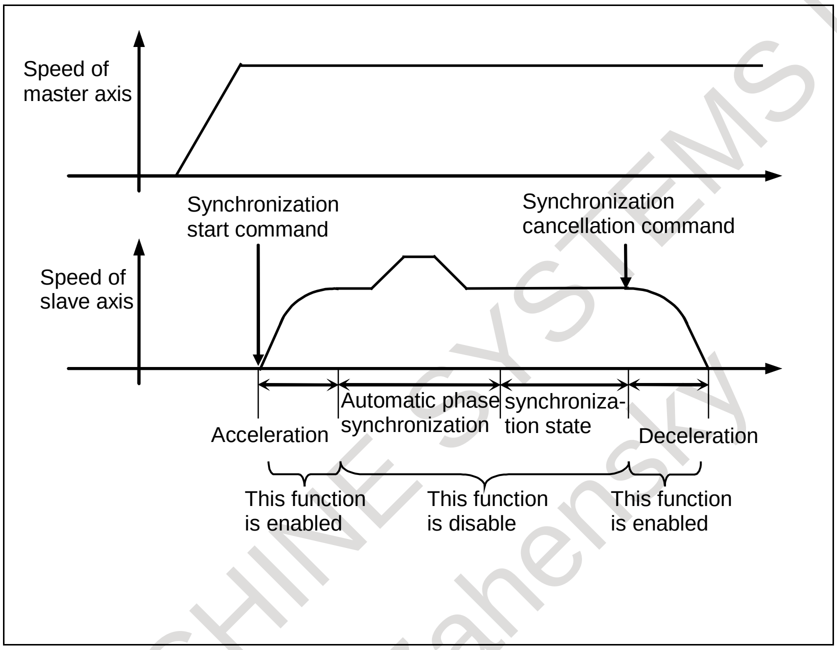
Acceleration in higher speed area than parameter No.5697

The acceleration decreases in inverse proportion to the speed. The acceleration is as follows:

$$\text{Acceleration} = \text{Parameter No.7778} \times \frac{\text{Parameter No.5697}}{\text{Current speed}}$$

NOTE

1 To use this function, the axes related to the synchronization must be rotary axes A type.
2 If 0 or less is set to parameter No.7778, this function is disabled.
3 If 0 or less is set to parameter No.5697, constant acceleration (specified by parameter No.7778) is applied in all speed area.
4 This function is valid in acceleration/deceleration of synchronization start/cancellation. This function is invalid in acceleration/deceleration of automatic phase synchronization.



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5694						PAV		

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#2 PAV Variable acceleration function in automatic phase synchronization for flexible synchronization control is:
0: Disabled.
1: Enabled.

5697	Acceleration switch speed of slave axis in synchronization start/cancellation

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}

[Valid data range] 0 to 99999999

This parameter set the acceleration switch speed of slave axis in synchronization start/cancellation. If 0 or less is set to this parameter, constant acceleration (specified by parameter No.7778) is applied in all speed area.

> **NOTE**
> This parameter is valid If 1 is set to the parameter PAV(No.5694#2).

7778	Acceleration of slave axis in synchronization start/cancellation

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/ sec^2, deg/sec^2
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(For a millimeter machine, 0.0 to +100000.0, for an inch machine, 0.0 to +10000.0)

When variable acceleration function in automatic phase synchronization for flexible synchronization control is valid (PAV(No.5694#2)=1), this parameter sets acceleration of slave axis in synchronization start/cancellation. If 0 or less is set to this parameter, variable acceleration function in automatic phase synchronization for flexible synchronization control is disabled.

1.12.4 Synchronization Positional Difference Detection Diagnosis Display and Signal Output in Flexible Synchronization Control

Overview

This function displays the diagnosis data and outputs DO signal <Fn533> for confirming error of between master axis and slave axis after executing automatic phase synchronization for flexible synchronization control.

Diagnosis data

Error (parameters Nos. 5600 to 5603) and maximum error (parameters Nos. 5604 to 5607) between master axis and slave axis after executing automatic phase synchronization are displayed.

Output signal

If a positional error detected after automatic phase synchronization exceeds any of the settings of parameters Nos. 13437 to 13440 (threshold values for phase synchronization positional error detection signal output), the DO signal <Fn553> is output.

Signal

Automatic phase synchronization error detection signals PHERA, PHERB, PHERC, PHERD <Fn553.0-Fn553.3>

[Classification] Output signal

[Function] This signal can be used to check whether an automatic phase synchronization positional difference has exceeded the setting of parameters Nos. 13437 to 13440 (threshold values for automatic phase synchronization positional error detection signal output).

This signal becomes "1" when:

- Error of automatic phase synchronization excess threshold values.

This signal becomes "0" when:

- Error of automatic phase synchronization does not excess threshold values.

- Flexible synchronization control phase synchronization end signal PHFINA, PHFINB, PHFINC, PHFIND <Fn381.0-Fn381.3> is "0".

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn553					PHERD	PHERC	PHERB	PHERA

Parameter

13437	Threshold value for automatic phase synchronization error detection signal output (group A)
13438	Threshold value for automatic phase synchronization error detection signal output (group B)
13439	Threshold value for automatic phase synchronization error detection signal output (group C)
13440	Threshold value for automatic phase synchronization error detection signal output (group D)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.000 to +999999.999)

If error between master axis and slave axis after executing automatic phase synchronization for flexible synchronization control, automatic phase synchronization error detection signal PHERA, PHERB, PHERC, PHERD is turned "1".

Please set this parameter in the path of slave axis in inter-path flexible synchronization control.

Diagnosis data

5600	Error of automatic phase synchronization (group A)
5601	Error of automatic phase synchronization (group B)
5602	Error of automatic phase synchronization (group C)
5603	Error of automatic phase synchronization (group D)

[Data type] Real path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data
(When the increment system is IS-B, -999999.999 to +999999.999)

Error between master axis and slave axis after executing automatic phase Synchronization for flexible synchronization control is displayed.

This data is displayed in the path of slave axis in inter-path flexible synchronization control.

5604	Maximum error of Automatic Phase Synchronization (group A)
5605	Maximum error of Automatic Phase Synchronization (group B)
5606	Maximum error of Automatic Phase Synchronization (group C)
5607	Maximum error of Automatic Phase Synchronization (group D)

[Data type] Real path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data
(When the increment system is IS-B, -999999.999 to +999999.999)
Maximum error between master axis and slave axis after executing automatic phase synchronization for flexible synchronization control is displayed.
This data is displayed in the path of slave axis in inter-path flexible synchronization control.
This data is cleared when automatic operation is started in auto mode.
This data is cleared when flexible synchronization control is started in manual mode.

Notes

NOTE

Diagnosis data is displayed after the following time passes since flexible synchronization control phase synchronization end signal is turned "1".
1/ (Parameter No.1825 ∗ 0.01) ∗ 3 [sec]
Example)
If No. 1825 = 3000, diagnosis data is displayed 100 msec later.
This function is usable only when the gear ratio between the master and slave axes is 1 vs. 1.

1.12.5 Inter-path Flexible Synchronization Control

Overview

Inter-path flexible synchronization control enables flexible synchronization control between axes in different paths in a multi-path system.
Up to four slave axes can be specified in one path.
An axis in another path can be specified as the master axis of each slave axis.
Synchronization for all synchronization pairs in all paths can be executed simultaneously.

Example)
In a multi-path system with the following axis configuration (Fig. 1.12.5 (a)), not only synchronization between the C1 axis in path 1 (master axis) and the A1 axis in path 1 (slave axis), but also synchronization between the C1 axis in path 1 (master axis) and the A2 axis in path 2 (slave axis) can be performed.

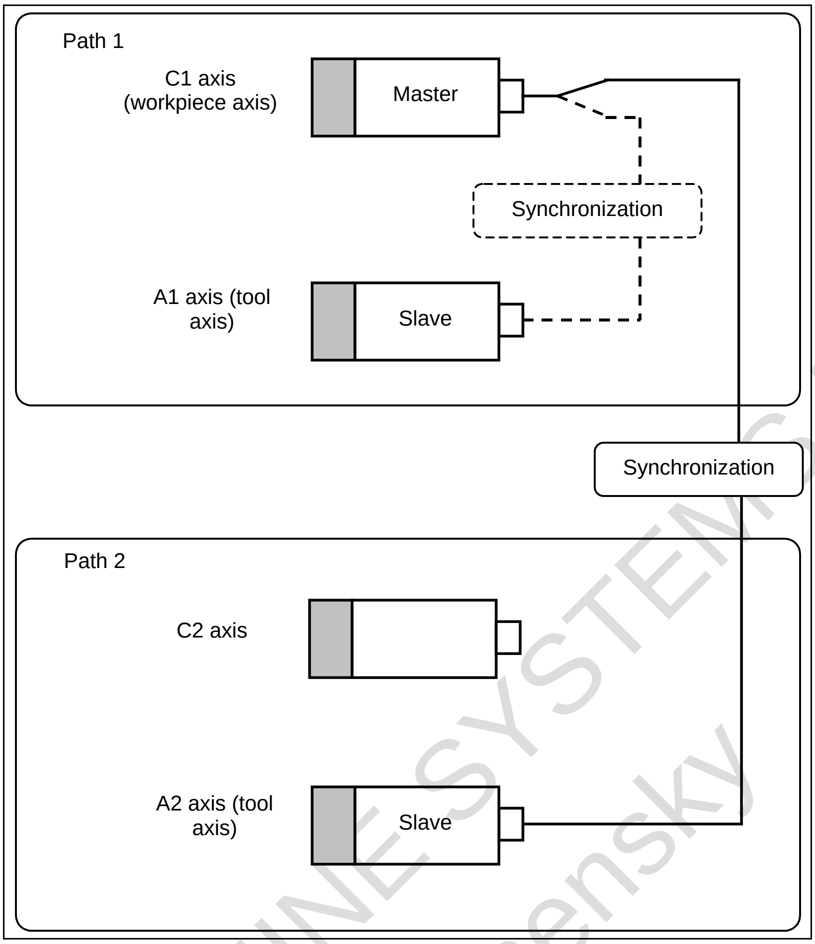


Fig. 1.12.5 (a)

Details

Setting of synchronization axes

Set master axis numbers in parameters Nos. 5660, 5662, 5664, and 5666.
Set slave axis numbers in parameters Nos. 5661, 5663, 5665, and 5667.
Set the number obtained by adding the controlled axis number to path number × 100 in these parameters for the slave axis path.

Example)
Master axis: 1st axis in path 1, slave axis: 1st axis in path 2
Parameter No. 5660 (path 2) = 101
Parameter No. 5661 (path 2) = 201

A slave axis in a synchronization group can be set as the master axis in another group.
Example)
The following synchronization is be applied:

$$\begin{tikzcd} X1 \arrow[r] & X2 \arrow[d] \\ Y1 & Y2 \arrow[l] \end{tikzcd}$$

X1 (master) → X2 (slave)
X2 (master) → Y2 (slave)
Y (master) → Y1 (slave)

If the relation among the master and slave axes makes a loop, however, alarm PS5242 "ILLEGAL AXIS NUMBER" is issued when synchronization starts.

Example)

The following synchronization cannot be applied:

$$\begin{tikzcd} X1 \arrow[r] & X2 \arrow[d] \\ Y1 & Y2 \arrow[ul] \end{tikzcd}$$

X1 (master) → X2 (slave)
X2 (master) → Y2 (slave)
Y2 (master) → X1 (slave)

X1 is defined as its own master axis.

Specification method

To execute inter-path flexible synchronization control during automatic operation, it is necessary to put the CNC in the inter-path flexible synchronization mode by setting the inter-path flexible synchronization mode select signal OVLN <Gn531.4> to "1" with a ladder program by specifying an M code in the paths to which the master and slave axes belong.

If flexible synchronization is started when the inter-path flexible synchronization mode is off, alarm PS5245 "OTHER AXIS ARE COMMANDED" is issued.
The inter-path flexible synchronization mode is disabled by setting OVLN to "0".

For the program sequence of the inter-path flexible synchronization mode status and M code commands for flexible synchronization control, see the following example.

Example)

Path of master axis	Path of slave axis
M210; ------ Turns the inter-path flexible synchronization mode on. M100P12; ------ Waiting (Note 1) ~	M310; ------ Turns the inter-path flexible synchronization mode on. M100P12; ------ Waiting (Note 1) ~
: ------ Master axis move commands	M60; ------ Starts inter-path flexible synchronization. ~ M61; ------ Cancels inter-path flexible synchronization.
~ M110P12; ------ Waiting (Note 1) M211; ------ Turns the inter-path flexible synchronization mode off.	~ M110P12; ----- Waiting (Note 1) M311; ------ Turns the inter-path flexible synchronization mode off.

AI contour control I/II can be turned on or off when both the master and slave paths are in the inter-path flexible synchronization mode.
AI contour control I/II can also be turned on or off even if PMC axis control is used for the master axis during flexible synchronization.

NOTE

1. Specify the waiting M code after the M code for turning the inter-path flexible synchronization mode on and before the M code for turning the inter-path flexible synchronization mode off during automatic operation.
2. The M code for turning the inter-path flexible synchronization mode on controls signals as follows:
   Sets the inter-path flexible synchronization mode select signal OVLN to "1" and confirms that the inter-path flexible synchronization mode signal OVLNS <Fn545.1> is changed to "1".
3. The M code for turning the inter-path flexible synchronization mode off controls signals as follows:
   Sets the inter-path flexible synchronization mode select signal OVLN to "0" and confirms that the inter-path flexible synchronization mode signal OVLNS is changed to "0".

Restrictions

The following functions cannot be specified in the inter-path flexible synchronization mode.
If any of these functions is specified in the inter-path flexible synchronization mode, alarm PS0502 "ILLEGAL G-CODE" is issued.

- Reference return in Cs contouring control (G00, G28)
- Skip function (G31)
- Automatic tool length measurement/Automatic tool offset function
- Automatic reference return operation of low-speed type (G28)
- High-speed program check function

These functions can be specified when flexible synchronization control and the inter-path flexible synchronization mode are turned off.

Inter-path flexible synchronization mode turned on immediately after power-on

When the inter-path flexible synchronization mode select signal is set to "1" immediately after power-on, the waiting M code does need to be specified.
For the program sequence of the inter-path flexible synchronization mode status and M code commands for flexible synchronization control, see the following example. (This example also explains the sequence when G31 is specified.)

Example)

Path of master axis	Path of slave axis
Set OVLN to 1. (The inter-path flexible synchronization mode is turned on.)	Set OVLN to 1. (The inter-path flexible synchronization mode is turned on.)
~	~
: ------ Master axis move commands	M60; ------ Starts inter-path flexible synchronization. ~ M61; ------ Cancels inter-path flexible synchronization.
M110P12; ------ Waiting M211; ------ Turns the inter-path flexible synchronization mode off. (NOTE 1)	M110P12; ----- Waiting M311; ------ Turns the inter-path flexible synchronization mode off. (NOTE 1)
G31_ ;	~
M210; ------ Turns the inter-path flexible synchronization mode on. (NOTE 1) M100P12; ------ Waiting	M310; ------ Turns the inter-path flexible synchronization mode on. (NOTE 1) M100P12; ------ Waiting
: ------ Master axis move commands	M60; ------ Starts inter-path flexible synchronization. ~ M61; ------ Cancels inter-path flexible synchronization.

NOTE

1. Turn the inter-path flexible synchronization mode on or off after flexible synchronization is canceled in the slave path.
2. Start or cancel flexible synchronization control with the signal (MTA, MTB, MTC, or MTD) in the slave path. Specify the M code for starting or canceling flexible synchronization in a block without specifying other commands.

NOTE

3 Flexible synchronization control cannot be started or canceled in the slave axis path while the path is in any of the following modes. In any of these modes, alarm PS5244, "TOO MANY DI ON" is issued.
   - Tilted working plane indexing

   Flexible synchronization control can be started or canceled in the master axis path, however.

4 Use an M code preventing buffering as the M code for turning the inter-path flexible synchronization mode on or off. (Parameter No. 3411 to 3420 or 11290 to 11299)
   Specify the M code for turning the inter-path flexible synchronization mode on or off in a block without specifying other commands.

5 Do not specify any move or PMC axis control command between the M code for turning the inter-path flexible synchronization mode on or off and a waiting M code.

6 An axis in spindle control with servo motor cannot be used as the master axis of flexible synchronization control.

Synchronization start
Command sequence

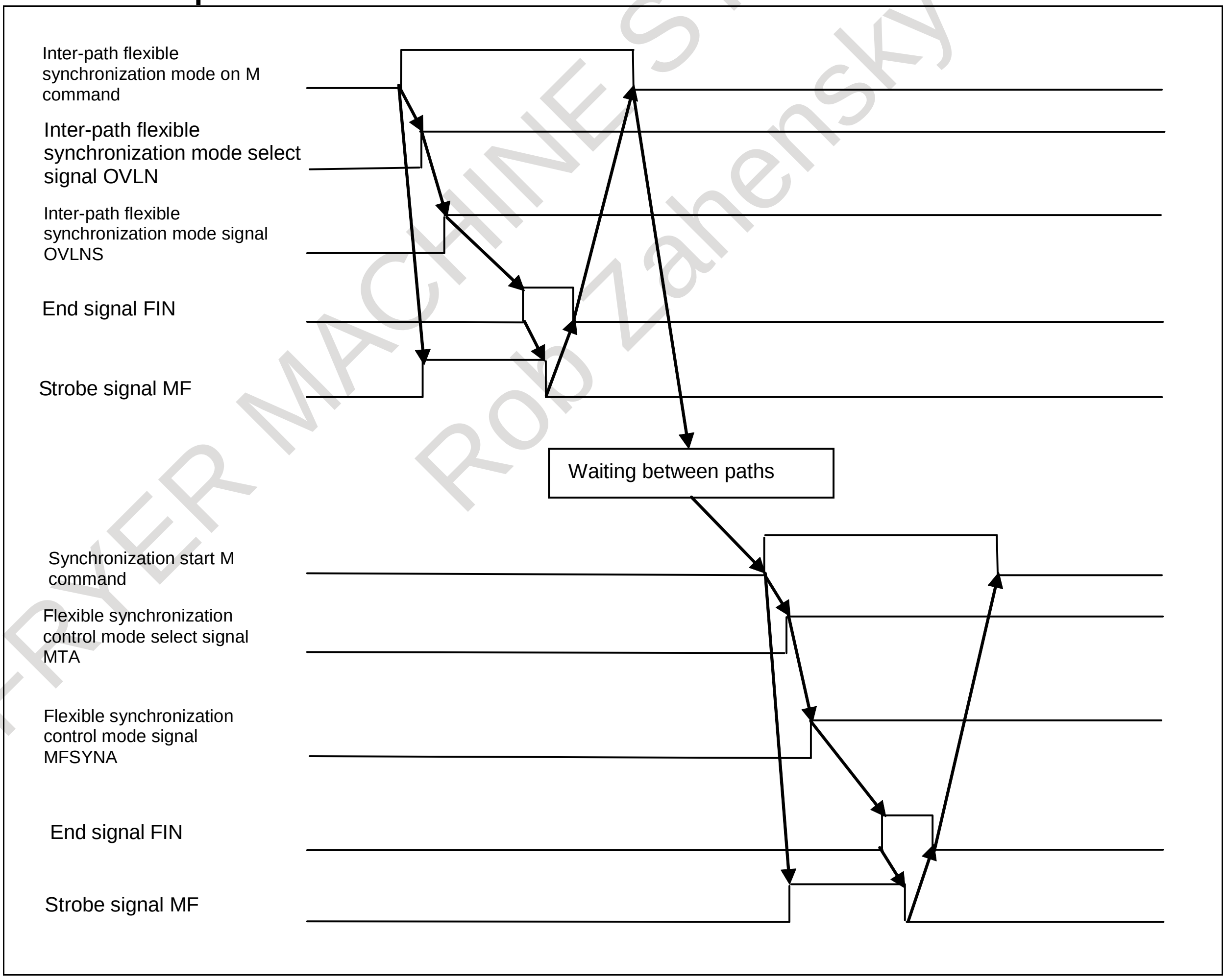


Fig. 1.12.5 (b)

1. When the M code for enabling the inter-path flexible synchronization mode is specified and the inter-path flexible synchronization mode select signal OVLN is set to "1", the inter-path flexible synchronization mode signal becomes "1".
2. The end signal FIN is operated for completion.
3. The waiting M code is specified in both the master and slave axis paths.
4. When the M code for enabling flexible synchronization control is specified, and the flexible synchronization control mode selection signal MTA, MTB, MTC, or MTD is set to "1", synchronization starts.
   And the flexible synchronization control mode status signal MFSYNA, MFSYNB, MFSYNC, or MFSYND becomes "1".
5. When the flexible synchronization control mode signal becomes "1", the end signal FIN is operated for completion.

Synchronization cancellation
Command sequence

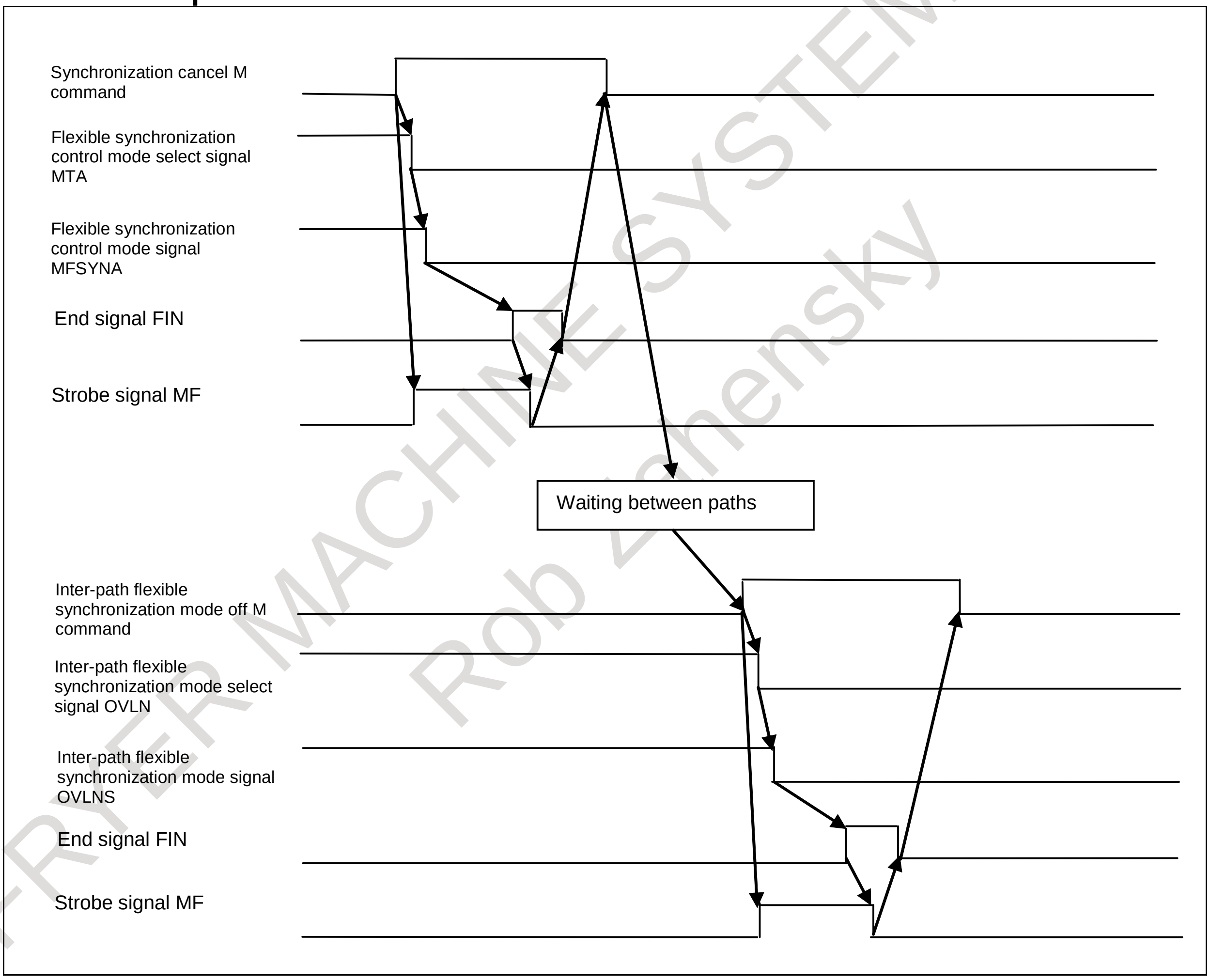


Fig. 1.12.5 (c)

1. When the M code for canceling flexible synchronization control is specified, and the flexible synchronization control mode selection signal is set to "0", synchronization is canceled.
2. When the flexible synchronization control mode status signal becomes "0", the end signal FIN is operated for completion.
3. The waiting M code is specified in both the master and slave axis paths.
4. When the M code for disabling the inter-path flexible synchronization mode is specified, and the inter-path flexible synchronization mode select signal OVLN is set to "0", the inter-path flexible synchronization mode is canceled.

5. When the inter-path flexible synchronization mode signal becomes 0, the end signal FIN is operated for completion.

Notes

NOTE

1 If an overtravel alarm occurs on the slave axis, the tool is stopped along both the master and slave axes, and alarm PS5245, "OTHER AXIS ARE COMMANDED" is issued in the master axis path.
2 If a servo alarm occurs, flexible synchronization control is canceled, and alarm PS5245 is issued in both the master and slave paths.
3 If an emergency stop is applied, flexible synchronization control is canceled and alarm PS5245 is issued in the path in which the emergency stop is not applied. When bit 1 (FCN) of parameter No. 1342 is 1, however, synchronization is not canceled and no alarm is issued.
4 Machine lock, if applied to the slave axis, is disabled while flexible synchronization is enabled.

Parameter

5660	Master axis number for flexible synchronization control (group A)
5661	Slave axis number for flexible synchronization control (group A)
5662	Master axis number for flexible synchronization control (group B)
5663	Slave axis number for flexible synchronization control (group B)
5664	Master axis number for flexible synchronization control (group C)
5665	Slave axis number for flexible synchronization control (group C)
5666	Master axis number for flexible synchronization control (group D)
5667	Slave axis number for flexible synchronization control (group D)

[Input type] Parameter input
[Data Input type] Word path
[Valid data range] 0 to Number of controlled axes
or m × 100+n (m:1 to Number of paths, n:1 to Number of controlled axes)
Specify both master and slave axis numbers.
Setting value)
1 to 24: Controlled axes on own path (one path system only)
101 to 124: Controlled axes in path 1
201 to 224: Controlled axes in path 2

In inter-path flexible synchronization control, an axis of a path can be specified as the master axis of another path.

NOTE

In inter-path flexible synchronization control, an axis of any path cannot be specified as the slave axis of another path.

	#7	#6	#5	#4	#3	#2	#1	#0
13421					FSV	FRF	FCN	

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> Set these parameters for the first path only. It will be effective to all paths.

#1 FCN In the emergency stop/servo off state, inter-path flexible synchronization control is:
0: Canceled.
1: Not canceled.

#2 FRF If G27/G28/G29/G30/G53 is specified during flexible synchronization control, alarm PS0010 is:
0: Issued.
1: Is not issued. Commands to the master axis are possible.
Even if, however, parameter bit 2 (FRF) is set to 1, and G28 is specified for the master axis in the state in which the reference position of the master axis subject to flexible synchronization control is not established, or if G27/G28/G29/G30/G53 is specified for the slave axis, alarm PS5381 is issued.

> **NOTE**
> When Inter-path flexible synchronization control is valid, even if 0 is set in the parameter bit FRF, the operation will be the same as that if 1 is set.

#3 FSV When the axis related to synchronization is servo off state while flexible synchronization control or inter-path flexible synchronization control, an automatic operation is:
0: Stopped.
1: Stopped if the axis related to synchronization moves.

> **NOTE**
> In inter-path flexible synchronization control, this parameter becomes effective when parameter FCN (No.13421#1) is set to 1.

Signal

Inter-path flexible synchronization mode select signal OVLN<Gn531.4>

[Classification] Input signal
[Function] Inter-path flexible synchronization control is enabled during automatic operation when the inter-path flexible synchronization mode is enabled in the relevant paths.
[Operation] When this signal becomes "1", the control unit operates as follows:
- Enables the inter-path flexible synchronization mode in the relevant paths.

> **NOTE**
> 1 To switch the inter-path flexible synchronization mode select signal between "1" and "0", the tool must be stopped along all axes (other than PMC axes) in the target path. If the tool moves along any axis, alarm DS0071 "START OR RELEASE CANNOT BE DONE" is issued.

> **NOTE**
> 2 Be sure to specify the waiting M code following the M code for enabling the inter-path flexible synchronization mode during automatic operation.
> 3 Be sure to specify the waiting M code preceding the M code for disabling the inter-path flexible synchronization mode during automatic operation.

Inter-path flexible synchronization mode signal OVLNS<Fn545.1>

[Classification] Output signal

[Function] This signal indicates the inter-path flexible synchronization mode status.

[Operation] This signal becomes "1" when:

- The relevant path is in the inter-path flexible synchronization mode.

This signal becomes "0" when:

- The relevant path is not in the inter-path flexible synchronization mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn531				OVLN				

	#7	#6	#5	#4	#3	#2	#1	#0
Fn545							OVLNS	

Alarm and message

Number	Message	Description
PS0502	ILLEGAL G-CODE	A G code unavailable in the inter-path flexible synchronization mode was specified.
PS5242	ILLEGAL AXIS NUMBER	This alarm is issued when either of the following conditions is satisfied. (It is issued at the start of inter-path flexible synchronization control.) 1. The master or slave axis number is invalid. 2. The relation among master and slave axes makes a loop.
PS5244	TOO MANY DI ON	- When an attempt was made to change the flexible synchronization control status, the select signal was not turned on or off after the execution of the M code. - An attempt was made to turn flexible synchronization control on or off without stopping the tool along all axes. (Except when automatic phase synchronization for flexible synchronization control is used) - Flexible synchronization control was turned off in any of the following function modes: - Tilted working plane indexing
PS5245	OTHER AXIS ARE COMMANDED	This alarm is issued when one of the following conditions is satisfied: 1. An overtravel alarm occurred in the slave axis. 2. A servo alarm occurred in a path in inter-path flexible synchronization control. 3. An emergency stop was applied in another path in inter-path flexible synchronization control. 4. When an attempt was made to execute flexible synchronization between different paths during automatic operation, the inter-path flexible synchronization mode was not enabled.

Number	Message	Description
PS5381	INVALID COMMAND IN FSC MODE	Either of the following commands was specified: - G28 was specified for the master axis when the reference position was not established for the master axis of flexible synchronization control. - G27, G28, G29, G30, or G53 was specified for the slave axis.
DS0071	START OR RELEASE CANNOT BE DONE	To start or cancel the inter-path flexible synchronization mode, the tool must be stopped along all axes.

1.12.6 Skip Function for Flexible Synchronization Control

Outline

This function enables the skip or high-speed skip signal (in the following explanation, these signals are collectively called skip signal) for the slave axis that is moved by command of the master axis in the flexible synchronization control mode.
This function has features such as the following:

- If a skip signal is input while a skip command for flexible synchronization control block is being executed, this block does not terminate until the specified number of skip signals have been input.
- The machine coordinates assumed when skip signals are input and the number of input skip signals are stored in specified custom macro variables.
- The total number of the skip signal inputs is stored in another specified custom macro variable.

Format

Mxx ; **Flexible synchronization control mode on**

G31.8 G91 α0 P_ Q_ R_ ; **Skip command for flexible synchronization control**

α : Specify the slave axis. The instruction value must be 0. It is considered 0 even if numbers except 0 are specified.
P_ : The top number of the consecutive custom macro variables in which the machine coordinate positions of the slave axis at the skip signal inputs are stored.
Q_ : The maximum allowable number of the skip signal inputs. (Range of command value: 1 to 512)
R_ : The number of the custom macro variables in which the total number of the inputs is stored.
This data is usually the same as the value specified by Q.
Therefore this is not necessarily specified. Specify it to check the number of skip signal inputs.

G31.8 is a one-shot G code.
During the execution of the G31.8 block, the machine coordinate positions of the slave axis at the skip signal inputs are stored in the consecutive custom macro variables where the top number of the variables is specified by P and the maximum allowable number of the skip signal inputs is specified by Q.
Also, this total number of the skip signal inputs is stored in the variable specified by R.

Example)

```
Mxx                          Flexible synchronization control mode on
X--
Y--
G31.8 G91 A0 P100 Q30 R1     Skip command for flexible synchronization control
```

After 30 times of skip signal inputs, 30 machine coordinate positions of the A axis are stored respectively in the consecutive custom macro variables #100 to #129.
The total number of skip signal inputs is stored in the custom macro variable #1.

NOTE

1. In the G31.9 or G31.8 block, when the axis was not specified, the alarm (PS1152) "G31.9/G31.8 FORMAT ERROR" occurs.
2. In the G31.8 block, only one slave axis should be commanded. When more than two slave axes are specified, the alarm (PS1152) occurs.
3. If G31.8 is commanded out of the flexible synchronization control mode (Flexible synchronization control mode status signal switching accepted signal (MFSYNA, MFSYNB, MFSYNC or MFSYND) is “0”), the alarm (PS1152) occurs.
4. If P is not specified, the alarm (PS1152) occurs.
5. If the axis besides a slave axis is specified, the alarm (PS1152) occurs.
6. If Q is specified out of range, the alarm (PS1152) occurs.
7. If P is not specified, the alarm (PS1152) occurs.
8. If R is not specified, the number of input skip signals is not written to a custom macro variable.
9. The number of custom macro variables specified in P and R must be the existing ones. If any nonexistent variable is specified, the alarm (PS0115) "VARIABLE NO. OUT OF RANGE" occurs. If a variable shortage occurs, the alarm (PS0115) occurs, too.
10. Whether to use conventional skip signals or high-speed skip signals with this function can be specified with the bit 4(HSS) of parameter No.6200. When high-speed skip is selected, specify which high-speed signals to enable with setting parameter 9S1 to 9S8 (No.6208#0 to #7).
11. The accumulated pulses and positional deviation due to acceleration/deceleration is considered and compensated when storing the machine coordinate positions to the custom macro variables.

Signal

- Please refer about signals of skip and flexible synchronization control to the following specification. ”Skip function” and “Flexible synchronization control” of “FANUC Series 0*i*-F Plus CONNECTION MANUAL (FUNCTION) (B- 64693EN-1)”.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200		SRE	SLS	HSS			SK0	GSK

[Input type] Parameter input
[Data type] Bit path

#0 GSK As a skip signal, the skip signal SKIPP is
0: Invalid.
1: Valid.

#1 SK0 This parameter specifies whether the skip signal is made valid under the state of the skip signal SKIP and the multistage skip signals SKIP2 to SKIP8.
0: Skip signal is valid when these signals are "1".
1: Skip signal is valid when these signals are "0".

#4 HSS 0: The skip function does not use high-speed skip signals while skip signals are input. (The conventional skip signal is used.)
1: The step skip function uses high-speed skip signals while skip signals are input.

#5 SLS 0: The multi-step skip function does not use high-speed skip signals while skip signals are input.
(The conventional skip signal is used.)
1: The multi-step skip function uses high-speed skip signals while skip signals are input.

> **NOTE**
> The skip signals (SKIP and SKIP2 to SKIP8) are valid regardless of the setting of this parameter. They can also be disabled using bit 4 (IGX) of parameter No. 6201.
> If you want to use high-speed skip signals when the multi-step skip function is used, set this parameter to 1.

#6 SRE When a high-speed skip signal is used:
0: The signal is assumed to be input on the rising edge (contact open → close).
1: The signal is assumed to be input on the falling edge (contact close → open).

	#7	#6	#5	#4	#3	#2	#1	#0
6201	**SKPXE**		**CSE**	**IGX**				

[Input type] Parameter input
[Data type] Bit path

#4 IGX When the high-speed skip function is used, SKIP, SKIPP, and SKIP2 to SKIP8 are:
0: Enabled as skip signals.
1: Disabled as skip signals.

#5 CSE For the continuous high-speed skip command, high-speed skip signals are:
0: Effective at either a rising or falling edge (depending on the setting of bit 6 (SRE) of parameter No.6200)
1: Effective at both the rising and falling edges.

#7 SKPXE For the skip function (G31), the skip signal SKIP is:
0: Enabled.
1: Disabled.

Whether the skip signals are enabled or disabled

Parameter	Bit 4 (IGX) of parameter No. 6201	Bit 0 (GSK) of parameter No. 6200	Bit 7 (SKPXE) of parameter No .6201	Skip signal SKIPP	Skip signal SKIP	Multistage skip signals SKIP2-SKIP8
Setting	0	0	0	Disabled	Enabled	Enabled
	0	1	0	Enabled	Enabled	Enabled
	0	0	1	Disabled	Disabled	Enabled
	0	1	1	Enabled	Disabled	Enabled
	1	0	0	Disabled	Disabled	Disabled
	1	1	0	Disabled	Disabled	Disabled
	1	0	1	Disabled	Disabled	Disabled
	1	1	1	Disabled	Disabled	Disabled

Bit 4 (IGX) of parameter No. 6201 is valid for the skip function using high-speed skip signals (when bit 4 (HSS) of parameter No. 6200 is set to 1) or for the multi-step skip function using high-speed skip signals (when bit 5 (SLS) of parameter No. 6200 is set to 1).

	#7	#6	#5	#4	#3	#2	#1	#0
6208	9S8	9S7	9S6	9S5	9S4	9S3	9S2	9S1

[Input type] Parameter input
[Data type] Bit path

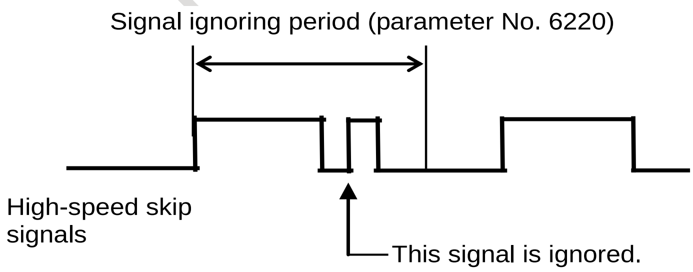
9S1 to 9S8 Specify which high-speed skip signal is enabled for the continuous high-speed skip command G31P90 or the EGB skip and the skip function for flexible synchronization control command G31.8.
The settings of each bit have the following meaning:
0: The high-speed skip signal corresponding to the bit is disabled.
1: The high-speed skip signal corresponding to the bit is enabled.
The bits correspond to signals as follows:

Parameter	High-speed skip signal	Parameter	High-speed skip signal
9S1	HDI0	9S5	HDI4
9S2	HDI1	9S6	HDI5
9S3	HDI2	9S7	HDI6
9S4	HDI3	9S8	HDI7

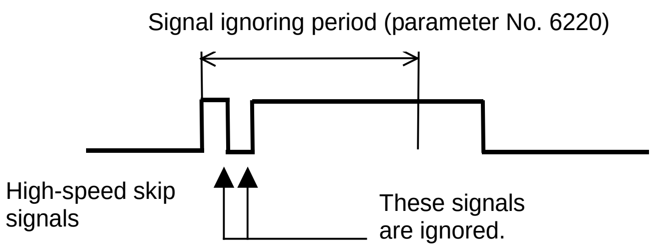
6220	Period during which skip signal input is ignored for the continuous high-speed skip function , EGB axis skip function , and flexible synchronization control skip function

[Input type] Parameter input
[Data type] Byte path
[Unit of data] 8msec
[Valid data range] 3 to 127(× 8msec)
This parameter specifies the period from when a high-speed skip signal is input to when the next high-speed skip signal can be input for the continuous high-speed skip function, EGB axis skip function, and flexible synchronization control skip function.
This parameter is used to ignore chattering in skip signals.
If a value that falls outside the valid range is specified, the setting is assumed to be 24 msec.

Example 1) When bit 5 (CSE) of parameter No. 6201 is 0 and bit 6 (SRE) of parameter No. 6200 is 0:



Example 2) When bit 5 (CSE) of parameter No. 6201 is 1:



Alarm and message

Number	Message	Description
PS0115	VARIABLE NO. OUT OF RANGE	A number that cannot be used for a local variable, common variable, or system variable in a custom macro is specified. In the EGB axis skip function or skip function for flexible synchronization control (G31.8), a non-existent custom macro variable number is specified. Or, the number of custom macro variables used to store skip positions is not sufficient.
PS1152	G31.9/G31.8 FORMAT ERROR	G31.8 is commanded out of the flexible synchronization control mode. The format of the G31.9(continuous high-speed skip function) or G31.8(EGB skip function / skip function for flexible synchronization control) block is erroneous in the following cases: - The axis was not specified in the G31.9 or G31.8 block. - Multiple axes were specified in the G31.9 or G31.8 block. - The P code was not specified in the G31.9 or G31.8 block. - G31.8 was commanded the axis besides a slave axis in the flexible synchronization control mode. - The Q command of G31.8 block is out of range in flexible synchronization control mode.

1.12.7 Hob Command by Flexible Synchronization Control

Overview

The same hob command as the electronic gearbox can be used as a synchronization command of flexible synchronization control.

Format

	Bit 0 (EFX) of parameter No.7731=0	Bit 0 (EFX) of parameter No.7731=1	
		Bit 5 (HBR) of parameter No.7731=1	Bit 5 (HBR) of parameter No.7731=0
Start of synchronization	G81 T_ (L_) (I_) (Q_ P_) ;	G81.4 R_ (L_) (I_) (Q_ P_) ;	G81.4 T_ (L_) (I_) (Q_ P_) ;
Cancellation of synchronization	G80 ;	G80.4 ;	G80.4 ;
	(*1) (*4)	(*2) (*4)	(*3) (*4)

T(or R) : Number of teeth (Specifiable range: 1 to 5000)

L : Number of hob threads (Specifiable range: -250 to 250)
The sign of L determines the direction of rotation for a slave axis.
When L is positive, the direction of rotation for a slave axis is positive (+ direction).
When L is negative, the direction of rotation for a slave axis is negative (- direction).
When L is 0, it follows the setting of bit 3 (LZR) of parameter No.7701.
If L is not specified, the number of hob threads is assumed 1.

Q : Module or diametrical pitch
Specify a module in the case of metric input.
(Unit: 0.00001mm, Specifiable range: 0.01 to 25.0mm)
Specify a diametrical pitch in the case of inch input.
(Unit: 0.00001inch^{-1}, Specifiable range: 0.01 to 254.0 inch^{-1})

P : Gear helix angle
(Unit: 0.0001deg, Specifiable range: -90.0 to 90.0deg)

I : The number of a group starting flexible synchronization control
(Group A = 1, B = 2, C = 3, D = 4)
If I is not specified, the group is assumed A.

*1 Use it for machining center system.
*2 Use it for lathe system.
*3 Use it for machining center system.
*4 When processing a helical gear, it specifies both Q and P. A decimal point can be used when specifying Q and P.

NOTE
Specify G81, G80, G81.4, and G80.4 in separate blocks.

Explanation

In conventional, flexible synchronization control has been started synchronization by setting flexible synchronization control mode selection signals MTA, MTB, MTC and MTD <Gn197.0, Gn197.1, Gn197.2 and Gn197.3> to "1". In this function, a command compatible with that for a hobbing machine can be used as a synchronization command of flexible synchronization control. This function is enabled by setting bit 0 (HOB) of parameter No.5694 to 1.

NOTE
If bit 0 (HOB) of parameter No.5694 is set to 1, the function related to "Electronic gear box" cannot be used.

Example of programming)
Group A : The master axis is B, the slave axis is C, and gear ratio is 1 : 50.
Group B : The master axis is Z, the slave axis is C, and gear ratio is 1 : 5.
Group C : The master axis is Y, the slave axis is C, and gear ratio is 23 : 20.

G81 T1 L50 I1 ; .... B - C synchronization start
G81 T1 L5 I2 ; .... Z - C synchronization start
:
G81 T23 L20 I3 ; .... Y - C synchronization start
:
G80 ; .... All synchronization cancellation
:

- Synchronization start

Specify P and Q to use helical gear compensation.
In this case, if only either of P and Q is specified, alarm PS0544 "FSC FORMAT ERROR" is issued. When G81 is specified so that the machine enters synchronization mode, the synchronization of a master axis and a slave axis is started.
During synchronization, control is performed such that the ratio of the master axis speed to the slave axis speed is the same as that of T (number of teeth) to L (number of hob threads).
During synchronization, if G81 is specified again without synchronization cancellation, it is changed to the one newly specified with T, L, P and Q commands.
It is possible that two or more groups are synchronization mode at same time as the above-mentioned example of programming.

- Synchronization cancellation

The synchronization of all synchronized groups is canceled with G80.
Absolute position display and relative position display for a slave axis are updated by adding the amount of movement due to the synchronization in synchronization cancellation. If the group is not updated machine position display by setting bit 0 to 3 (ACA to ACD) parameter No.5668 to 1, the above-mentioned absolute position display and relative position display are not updated.
In a G80 block, do not specify addresses other than O or N.

NOTE

1 If bit 0 (EFX) of parameter No. 7731 is 0, no canned cycle for drilling can be used. To use a canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.
2 If bit 0 (TDP) of parameter No. 7702 is 1, the permissible range of T is 0.1 to 300 (1/10 of the specified value).
3 If, at the start of flexible synchronization control (G81), L is specified as 0, the synchronization starts with L assumed to be 1 if bit 3 (LZR) of parameter No.7701 is 0; if bit 3 (LZR) of parameter No.7701 is 1, the synchronization is not started with L assumed to be 0. At this time, helical gear compensation is performed.

- Synchronization coefficient

Synchronization coefficient of flexible synchronization control by hob command is calculated from the following equation.

Synchronization coefficient = $L/T \times \beta/\alpha$

T: Number of teeth
L: Number of hob threads

α: Pulse count of position detector per rotation about a master axis (Parameter No.7782)
β: Pulse count of position detector per rotation about a slave axis (Parameter No.7783)

- Inter-path flexible synchronization control

Start and cancellation by hob command is effective for the group of flexible synchronization control to which belongs to a path specified. In inter-path flexible synchronization control, it specifies in the path to which a slave axis belongs.

Example of programming)

The path to which a master axis belongs	The path to which a slave axis belongs
:	:
M210 ; .... Inter-path flexible synchronization control mode on	M310 ; .... Inter-path flexible synchronization control mode on
M100 P12 ; . Waiting	M100 P12 ; .... Waiting
:	G81 T1 L50 I1 ; . Flexible synchronization control start
:	:
:	:
:	G80 ; .... Flexible synchronization control cancellation
:	
M110P 12 ; . Waiting	M110 P12 ; .... Waiting
M211 ; .... Inter-path flexible synchronization control mode off	M311 ; .... Inter-path flexible synchronization control mode off
:	::

NOTE

1 Specify a waiting M code after the M code for turning inter-path flexible synchronization mode on and before the M code for turning inter-path flexible synchronization mode off during automatic operation.
2 In the M code for turning inter-path synchronization mode on, it sets inter-path flexible synchronization mode select signal OVLN to "1" and confirms that inter-path flexible synchronization mode signal OVLNS is changed to "1".
3 In the M code for turning inter-path synchronization mode off, it sets inter-path flexible synchronization mode select signal OVLN to "0" and confirms that inter-path flexible synchronization mode signal OVLNS is changed to "0".
4 Use the M code preventing buffering as the M code for turning inter-path flexible synchronization mode on or off.
(Parameter Nos. 3411 to 3420 or 11290 to 11299)
5 In the others note or limitation, refer to the specification of "Inter-path flexible synchronization control".

- Flexible synchronization control in case a master axis is controlled by "Spindle control with servo motor or PMC axis"

If PMC axis or spindle control with servo motor is a master axis, be sure to set the controlled axis selection signal EAXn<Gn136> for PMC axis control to "1" or turn on the SV speed mode of spindle control with servo motor before turning on the synchronization mode.
By setting bit 4 (FPA) of parameter No.13421 to 1, it is possible to issue the alarm PS5381 “INVALID COMMAND IN FSC MODE” when flexible synchronization control is operated in the incorrect procedure.

Example of programming)
:
S0 P11 ;
G96.4 P11 ; .... : SV speed control mode ON (In case of PMC axis,

```
                                                  EAXn<Gn136> is set to "1".)
G81 T1 L0 I1 ; ............................................. : Flexible synchronization control ON
S100 P11 ;
:
:
S0 P11 ;
G80 ; ............................................................ : Flexible synchronization control OFF
G96.1 P11 ; ................................................... : SV speed control mode OFF (In case of PMC axis,
                                                  EAXn<Gn136> is set to "0".)
:
```

NOTE

1 For a synchronization group for which PMC axis or spindle control with servo motor is a master axis, be sure to turn on the synchronization mode first. Also, for a synchronization group for which PMC axis or spindle control with servo motor is a master axis, be sure to turn off the synchronization mode last.
2 If an attempt is made to turn on a synchronization group for which PMC axis or spindle control with servo motor is a master axis when there exists a synchronization group for which non-PMC or non-spindle control with servo motor, normal axis is a master axis, alarm PS5245 "OTHER AXIS ARE COMMANDED" is issued.
3 In the others note or limitation, refer to the specification of "Flexible synchronization control".

When a master axis of flexible synchronization control is the axis of spindle control with servo motor, feed per revolution can be performed based on the rotation compensated with a gear ratio of a slave axis of flexible synchronization control.
Rotation speed that is used for the slave axis is shown as follows.

$$\text{Rotation compensated with a gear ratio of a slave axis} = \text{Feedback pulse of a position coder selected} \times \frac{L}{T}$$

T : Number of hob threads

L : Number of teeth

⚠ CAUTION

1. In case that all of following conditions are satisfied, feed per revolution is executed by the feedback pulse of position coder.
   - A group of flexible synchronization control set with parameter No.5695 is under synchronization mode.
   - A master axis is the axis of spindle control with servo motor.
   - The rotation of the master axis is selected as the feedback pulse.
2. The rotation compensated with a synchronization coefficient used by feed per revolution is not included the superimposed pulse from other master axes and the pulse instructed oneself.
3. Select the rotation of a master axis as the feedback pulse before turning on flexible synchronization control mode.

- Automatic phase synchronization

It is performed with operating signal as well as conventional. Automatic phase synchronization for flexible synchronization control can be used. Please show the specification of "Automatic phase synchronization for flexible synchronization control".

> ⚠ **CAUTION**
> The group of flexible synchronization control by hob command is changed the position for automatic phase synchronization of the slave axis by bit 6 (EPA) of parameter No.7731.

- Helical gear compensation

For a helical gear, a slave axis is subjected to compensation for movement along the Z axis (axial feed axis) according to the twisted angle of the gear.
Helical gear compensation is performed with the following data.

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (for metric input)}$$

$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (for inch input)}$$

Compensation angle: Absolute value with sign (degrees)
Z : Amount of travel along the Z axis after a G81 command is issued (mm or inch)
P : Twisted angle of the gear with sign (degrees)
π : Circular constant
T : Number of teeth
Q : Module (mm) or diametral pitch (inch-1)
Use P, T, and Q specified in the G81 block.

In helical gear compensation, the machine coordinates and the absolute coordinates on a slave axis are updated with helical gear compensation.

- Direction of helical gear compensation

The direction depends on bit 2 (HDR) of parameter No. 7700.

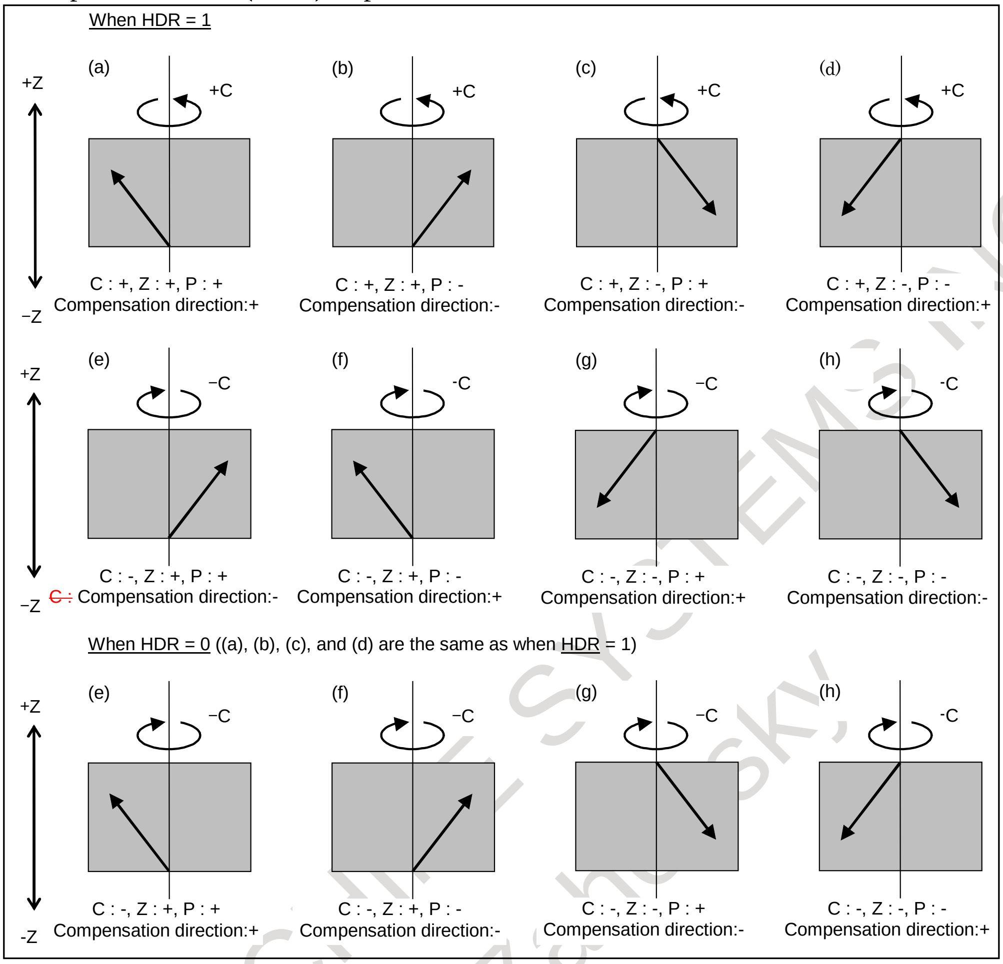


Fig. 1.12.7 Direction of helical gear compensation

- Acceleration/Deceleration of helical gear compensation

The time constant of acceleration/deceleration after interpolation of the slave axis is applied to the movement by helical gear compensation. Furthermore, when acceleration/deceleration before interpolation is valid, maximum allowable acceleration rate of the slave axis is not considered. If the acceleration of the movement by helical gear compensation is too large, reduce maximum allowable acceleration rate of the axial feed axis.

Note

- Set a master axis number and a slave axis number to the number of a master axis for flexible synchronization control (Parameters Nos.5660, 5662, 5664 and 5666) and the number of a slave axis for flexible synchronization control (Parameters Nos.5661, 5663, 5665 and 5667).
- It is necessary that a slave axis of flexible synchronization control that performs the hob command is set the type A of a rotation axis.
- If the M code number for turning on the flexible synchronization control mode is specified to a group of flexible synchronization control by hob command, alarm PS5381 "INVALID COMMAND IN FSC MODE" is issued.
- If hob command is specified to a group of flexible synchronization control by flexible synchronization control mode selection signals MTA, MTB, MTC and MTD, alarm PS5381 is issued.
- G81, G81.4, G80 and G80.4 used with hob command by flexible synchronization control prevent buffering.
- The synchronization mode by hob command is not canceled by reset operation.

- If G81 or G81.4 is re-specified to the group during the synchronization mode, it is necessary that automatic phase synchronization for flexible synchronization control is enabled.
- A synchronization group of turning on the flexible synchronization control mode by hob command is sure to turn on the synchronization mode first. Also, for a synchronization group of turning on the flexible synchronization control mode is sure to turn off the synchronization mode last. However, if the inter-path flexible synchronization mode select signal OVLN is set to "1", there is no restriction about the order. The inter-path flexible synchronization mode select signal OVLN can be used with flexible synchronization control in the path.
- The operation of synchronization mode that is not described in this specification is the same as turning on it by flexible synchronization control mode selection signals MTA, MTB, MTC and MTD. Please refer to the specification of "Flexible synchronization control" and "Inter-path flexible synchronization control".

Limitation

- If bit 0 (HOB) of parameter No.5694 is set to 1, the function related to electronic gear box cannot be used.
- In the group of turning on the flexible synchronization control mode by hob command, flexible synchronization control mode selection signals MTA, MTB, MTC and MTD is disabled.
- Inch/metric conversion commands (G20 and G21) cannot be specified in the synchronization mode by hob command.
- In the synchronization mode by hob command, AI contour control mode is temporarily canceled. However, when the inter-path flexible synchronization mode select signal OVLN is assumed to be "1", AI contour control I, AI contour control II and advanced preview feed forward of a slave axis can be made effective. Advanced preview feed forward of a slave axis is controlled by the mode or modal of a slave axis.
- In the program included hob command, "Program restart" and "Quick program restart" cannot be used.
- In "Retrace" and "Manual handle retrace", the blocks specified with G81, G81.4 are backward movement prohibition.
- This function cannot be used together with the function that cannot be used together with G code command (G81, G81.4, G80 and G80.4) of "Electronic gear box".

Signal

Inter-path flexible synchronization mode select signal OVLN<Gn531.4>

[Classification] Input signal

[Function] Inter-path flexible synchronization control is enabled during automatic operation when the inter-path flexible synchronization mode is enabled in the relevant paths.

[Operation] When this signal becomes "1", the control unit operates as follows:

- Enables the inter-path flexible synchronization mode in the relevant paths.

> **NOTE**
> 1 To switch the inter-path flexible synchronization mode select signal between "1" and "0", the tool must be stopped along all axes (other than PMC axes) in the target path. If the tool moves along any axis, alarm DS0071 "START OR RELEASE CANNOT BE DONE" is issued.
> 2 Be sure to specify the waiting M code following the M code for enabling the inter-path flexible synchronization mode during automatic operation.
> 3 Be sure to specify the waiting M code preceding the M code for disabling the inter-path flexible synchronization mode during automatic operation.

Flexible synchronization control mode status signals
MFSYNA, MFSYNB, MFSYNC, MFSYND <Fn197.0 to Fn197.3>

[Classification] Output signal

[Function] These signals are used to check that the groups selected with the flexible synchronization control mode selection signals are actually switched to that mode.

[Operation]
1) When the synchronization mode actually becomes effective to a group, the corresponding one of these signals is set to "1".
2) When the synchronization mode is actually canceled for a group, the corresponding one of these signals is set to "0".

> **NOTE**
> When the synchronization mode is switched by hob command, it becomes similar to the operation.

Inter-path flexible synchronization mode signal OVLNS<Fn545.1>

[Classification] Output signal

[Function] This signal indicates the inter-path flexible synchronization mode status.

[Operation] This signal becomes "1" when:
- The relevant path is in the inter-path flexible synchronization mode.

This signal becomes "0" when:
- The relevant path is not in the inter-path flexible synchronization mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn531				OVLN				
Fn197					MFSYND	MFSYNC	MFSYNB	MFSYNA
Fn545							OVLNS	

Parameter

The parameter used by this function is shown as follows.

5660	Master axis number for flexible synchronization control (group A)
5661	Slave axis number for flexible synchronization control (group A)
5662	Master axis number for flexible synchronization control (group B)
5663	Slave axis number for flexible synchronization control (group B)
5664	Master axis number for flexible synchronization control (group C)
5665	Slave axis number for flexible synchronization control (group C)
5666	Master axis number for flexible synchronization control (group D)
5667	Slave axis number for flexible synchronization control (group D)

[Input type] Parameter input

[Data Input type] Word path

[Valid data range] 0 to Number of controlled axes
or m × 100+n (m:1 to Number of paths, n:1 to Number of controlled axes)
Specify both master and slave axis numbers.
Setting value)
1 to 24: Controlled axes on own path (one path system only)

101 to 124: Controlled axes in path 1
201 to 224: Controlled axes in path 2

In inter-path flexible synchronization control, an axis of a path can be specified as the master axis of another path.

> **NOTE**
> In inter-path flexible synchronization control, an axis of any path cannot be specified as the slave axis of another path.

	#7	#6	#5	#4	#3	#2	#1	#0
5694								**HOB**

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 HOB The command of G80 and G81 or G80.4 and G81.4 is:
0: Used with "Electronic gear box".
1: Used with "Hob command by flexible synchronization control".

> **NOTE**
> It is selected whether to use G80 and G81 or G80.4 and G81.4 by bit 0 (EFX) of parameter No.7731.

5695	**Group number to which feed per revolution can be performed based on the rotation compensated with a synchronization coefficient of a slave axis of flexible synchronization control**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 4

The group number of flexible synchronization control which slave axis used with feed per revolution belongs is set.
Feed per revolution can be performed based on the rotation compensated with a synchronization coefficient of a slave axis of flexible synchronization control .
Set the group number of flexible synchronization control as follows.
Group A = 1, Group B = 2, Group C = 3, Group D = 4
If this parameter is set to 0, feed per revolution is performed based on the feedback pulse.

	#7	#6	#5	#4	#3	#2	#1	#0
7700						**HDR**		

[Input type] Parameter input
[Data type] Bit path

#2 HDR Direction of helical gear compensation (usually, set 1.)
(Example) To cut a left-twisted helical gear when the direction of rotation about the C-axis is the negative (-) direction:
0: Set a negative (-) value in P.
1: Set a positive (+) value in P.

	#7	#6	#5	#4	#3	#2	#1	#0
7701					LZR			

[Input type] Parameter input
[Data type] Bit path

#3 LZR When L (number of hob threads) = 0 is specified at the start of flexible synchronization control (G81):
0: Synchronization is started, assuming that L = 1 is specified.
1: Synchronization is not started, assuming that L = 0 is specified. However, helical gear compensation is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
7702								TDP

[Input type] Parameter input
[Data type] Bit path

#0 TDP The specifiable number of teeth, T, of flexible synchronization control (G81) is:
0: 1 to 5000
1: 0.1 to 500 (1/10 of a specified value)

> **NOTE**
> In either case, a value from 1 to 5000 can be specified.

7709	Number of the axial feed axis for helical gear compensation

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to Number of controlled axes

This parameter sets the number of the axial feed axis for a helical gear.

> **NOTE**
> When this parameter is set to 0 or a value outside the valid setting range, the Z-axis becomes the axial feed axis.

	#7	#6	#5	#4	#3	#2	#1	#0
7731	HAD	EPA	HBR					EFX

[Input type] Parameter input
[Data type] Bit path

#0 EFX Hob command by flexible synchronization control:
0: G80 and G81 are used.
1: G80.4 and G81.4 are used.

> **NOTE**
> When this parameter is set to 0, no canned cycle for drilling can be used.

#5 HBR In flexible synchronization control start command G81.4, the number of teeth is:
0: Specified in T.
1: Specified in R.

> **NOTE**
> This parameter is valid when bit 0 (EFX) of parameter No. 7731 is set to 1.

#6 EPA Automatic phase synchronization for flexible synchronization control by hob command is performed in such a way that:
0: The machine coordinate 0 of the slave axis is aligned to the machine coordinate 0 of the master axis.
1: The position of the slave axis at synchronization start is aligned to the machine coordinate 0 of the master axis.

#7 HAD In the absolute coordinate, the amount of helical gear compensation is updated:
0: When the synchronization is canceled.
1: During executing helical gear compensation.

7782	Number of pulses from the position detector per flexible synchronization control master axis rotation

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 999999999

For a slave axis, set the number of pulses generated from the position detector per flexible synchronization control master axis rotation. For an A/B phase detector, set this parameter with four pulses equaling one A/B phase cycle.

> **NOTE**
> This parameter is enabled when hob command is specified.

7783	Number of pulses from the position detector per flexible synchronization control slave axis rotation

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 999999999

For a slave axis, set the number of pulses generated from the position detector per flexible synchronization slave axis rotation. Set the number of pulses output by the detection unit.
All the settings of parameters Nos. 7782 and 7783 have to do is to indicate the ratio correctly. So, you can reduce the fraction indicated by the settings.

> **NOTE**
> This parameter is enabled when hob command is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
13421				FPA				

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> Set these parameters for the first path only. It will be effective to all paths.

#4 FPA For a synchronization group for which a PMC axis is a master axis, when the controlled axis selection signal EAXn<Gn136> for PMC axis control is set to "1" after starting flexible synchronization control mode, and the master axis is specified by PMC axis control, or for a synchronization group for which spindle control with servo motor is a master axis, when SV speed control mode of the master axis is turned on after starting flexible synchronization control mode:

0: Alarm is not generated.
1: Alarm PS5381 ”INVALID COMMAND IN FSC MODE” is generated.

This function generates the alarm when flexible synchronization control is operated by in the incorrect procedure.

> ⚠**CAUTION**
> When this parameter is set to 0, it operates without generating the alarm even if flexible synchronization control mode is started according to a procedure not correct. However, the position shift of a slave axis may occur.

Alarm and message

Number	Message	Description
PS0543	FSC PARAMETER SETTING ERROR	- The slave axis specified with G81 is not set as a rotation axis. (bit 0 (ROTx) of parameter No. 1006) - Number of pulses per rotation (parameter No. 7782 or 7783) is not set. - The group number of flexible synchronization control which slave axis used with feed per revolution belongs is not correctly set.
PS0544	FSC FORMAT ERROR	Error in the format of the block of hob command flexible synchronization control - T (number of teeth) is not specified in the G81 block. - In the G81 block, the data specified for one of T, L, P, and Q is out of its valid range. - In the G81 block, only one of P and Q is specified.
PS5243	DATA OUTRANGE	An overflow occurred in the calculation of the synchronization coefficient in hob command by flexible synchronization control.
PS5244	TOO MANY DI ON	In the program included hob command by flexible synchronization control (G81, G81.4, G80 and G80.4), the synchronization mode was not correctly switched when "Program restart" or "Quick program restart" was executed.
PS5245	OTHER AXIS ARE COMMANDED	An attempt was made to issue the following commands: An attempt was made to turn on a synchronization group for which an PMC axis or spindle control with servo motor was a master axis or the synchronization by hob command when there existed a flexible synchronization control group by the signal for which a non-PMC or non-spindle control with servo motor, normal axis was a master axis.

Number	Message	Description
PS5381	INVALID COMMAND IN FSC MODE	An attempt was made to issue the following commands: - An attempt was made to specify a M code for turning on or off flexible synchronization control to a synchronization group of turned on flexible synchronization control mode by hob command. - Hob command was specified to a synchronization group for which the synchronization mode was turned on with flexible synchronization mode selection signals MTA, MTB, MTC and MTD. - For a synchronization group for which a PMC axis is a master axis, when the controlled axis selection signal EAXn<Gn136> for PMC axis control was set to "1" after starting flexible synchronization control mode, and the master axis was specified by PMC axis control. - For a synchronization group for which spindle control with servo motor was a master axis, when SV speed control mode of the master axis was turned on after starting flexible synchronization control mode.

1.12.8 Hob Machining Package

Overview

Hob machining package is added.

Hob machining package enables to switch whether Electronic gear box or Flexible synchronization control is used for hob machining without restarting CNCs. With this function, you can evaluate both functions easily, and switch function to be used according to machining parts.

> **NOTE**
> - For details of Electronic gear box, refer to "ELECTRONIC GEAR BOX" in this manual.
> - For details of Flexible synchronous control, refer to "FLEXIBLE SYNCHRONIZATION CONTROL" in this manual.
> - For using a spindle as the master of hob machining with Hob command by flexible synchronization control, the spindle need to be switched to the Cs contour mode.
>   For details of Cs contour control, refer to "Cs CONTOUR CONTROL" in this manual.

Explanation

Hob machining package enables to switch whether Electronic gear box or Flexible synchronization control is used when a hob command is specified without restarting CNCs.

Conventionally, you can select which function is used for hob machining by changing the bit 0 (HOB) of parameter No.5694, and restarting the CNC is needed when it is changed. When Hob machining package is enabled, you can use the different function from the function selected by the bit 0 (HBS) of parameter No.5698, which can be changed without restarting the CNC, for hob machining.

The step to switch which function to be used for hob machining is as follows.

(1) Change CNC status to switch hob command. (refer to NOTE)
(2) Request to switch hob command by changing the bit 0 (HBS) of parameter No.5698.
(3) Confirm completion of switching hob command by checking the bit 0 (HOB) of diagnosis data No.5040.

Table 1.12.8 (a) Function to be used for hob machining when Hob machining package is enabled

parameter HOB(No.5694#0)	parameter HBS(No.5698#0)	Function to be used for hob machining
0	0	Electronic gear box
0	1	Flexible synchronization control
1	0	Flexible synchronization control
1	1	Electronic gear box

NOTE

1 If Hob machining package is enabled, hob commands (G80/G81 or G80.4/G81.4) are the G code for suppressing buffering.
2 When the bit 0 (HBS) of parameter No.5698 is changed, the both of the following conditions need to be met.
   - EGB synchronous control in the path is cancelled.
     (If Electronic gear box 2 pair is used, all pairs need to be cancelled.)
   - Flexible synchronization control in the path is cancelled.
     (All groups need to be cancelled.)
3 If the bit 0 (HBS) of parameter No.5698 is changed when the upper conditions are not met, switching hob command is executed after the conditions are met.
4 Whether switching hob command is finished or not can be confirmed with the bit 0 (HOB) of diagnosis data No.5040.

Restrictions

If the hob command switches without restarting the CNC, there are restrictions on the system configuration.
Examples of system configuration which can switch hob command without restarting CNC are shown as follows. (Assume that 1 path system is used)

(1) Both of the master spindle and the slave axis are the same after hob command is switched
Both of a Cs contour control axis and an EGB dummy axis are needed. In case of hob command by Flexible synchronization control, the master spindle needs to be switched to Cs contour mode. In case of hob command by Electronic gear box, Cs contour mode of the master spindle needs to be cancelled.

- Number of control axis is 6 (includes 2 virtual axes), Number of control spindle is 1

Linear axis	: X, Y, Z
Rotary axis	: A (Slave axis of EGB and FSC)
EGB dummy axis	: B
Cs contour axis	: C (Master axis of FSC)
Spindle	: S (Master axis of EGB and FSC)

(2) The master axis of Flexible synchronization control is a servo axis (Cs contour control isn't needed)
EGB dummy axis is needed. In this case, the master axis changes after a hob command is switched.

- Number of control axis is 6 (includes a virtual axis), Number of control spindle is 1

Linear axis	: X, Y, Z
Rotary axis	: A (The slave axis of EGB and FSC) C (The master axis of FSC)
EGB dummy axis	: B
Spindle	: S (The master axis of EGB)

To use the same master and slave axis between Electronic gear box and Flexible synchronization control, 2 virtual axes (EGB dummy axis and Cs contour axis) are needed to switch hob command without restarting the CNC.
If restarting the CNC is allowed when hob command switches, only a virtual axis (EGB dummy axis and Cs contour axis) is needed.

(3) Only a virtual axis is used (Restarting the CNC is needed to switch hob command)
The same master and slave axis between Electronic gear box and Flexible synchronization control can be used.

- Number of control axis is 5 (includes a virtual axis), Number of control spindle is 1

Linear axis	: X, Y, Z
Rotary axis	: A (The slave axis of EGB and FSC)
EGB dummy axis and Cs contour axis	: C (The master axis of FSC in case of Cs contour axis)
Spindle	: S (The slave axis of EGB and FSC)

> **NOTE**
> To switch the role of C axis (EGB dummy axis or Cs contour axis) with the upper system configuration of (3), parameter settings (parameter No.1023 and parameters related to FSSB) need to be changed.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5694								HOB

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 HOB The command of G80 and G81 or G80.4 and G81.4 is:
0: Used with “Electronic gear box”.
1: Used with “Hob command by flexible synchronization control”.

> **NOTE**
> 1 It is selected whether to use G80 and G81 or G80.4 and G81.4 by bit 0 (EFX) of parameter No.7731.
> 2 If Hob machining package is enabled and the bit 0 (HBS) of parameter No.5698 is 1, the different function from the function selected by this parameter is used.

	#7	#6	#5	#4	#3	#2	#1	#0
5698								HBS

[Input type] Parameter input
[Data type] Bit path

#0 HBS The command of G80 and G81 or G80.4 and G81.4 is:
0: Used with the function selected by the bit 0 (HOB) of parameter No.5694.
1: Used with the different function from the function selected by the bit 0 (HOB) of parameter No.5694.

NOTE
1 This parameter is enabled only when Hob machining package is enabled.
2 When this parameter is changed, the both of the following conditions need to be met.
  - EGB synchronous control in the path is cancelled.
    (If Electronic gear box 2 pair is used, all pairs need to be cancelled.)
  - Flexible synchronization control in the path is cancelled.
    (All groups need to be cancelled.)
3 If this parameter is changed when the upper conditions are not met, switching hob command is executed after the conditions are met.
4 Whether switching hob command is finished or not can be confirmed with the bit 0 (HOB) of diagnosis data No.5040.

	#7	#6	#5	#4	#3	#2	#1	#0
7731								EFX

[Input type] Parameter input
[Data type] Bit path

#0 EFX Hob command by flexible synchronization control:
0: G80 and G81 are used.
1: G80.4 and G81.4 are used.

NOTE
When this parameter is set to 0, no canned cycle for drilling can be used.

Alarm and message

Number	Message	Description
PW0000	POWER MUST BE OFF	A parameter was set for which the power must be turned OFF then ON again.

Diagnosis Data

	#7	#6	#5	#4	#3	#2	#1	#0
5040								HOB

#0 HOB G80/G81 or G80.4/G81.4 are :
0: used with Electronic gear box.
1: used with Flexible synchronization control.

NOTE
1 This is enabled when Hob machining package is enabled.
2 Confirm this to know which function is used for hob command, Electronic gear box or Flexible synchronization control. The different function from the function selected by parameter might be enabled.

1.13 AXIS IMMEDIATE STOP FUNCTION

Overview

When the movement long an axis must be immediately stopped, the axis immediate stop function stops the movement using the axis immediate stop start signal and outputs an alarm. In the AI contour control mode, this function changes the acceleration rate in look-ahead acceleration/deceleration before interpolation and stops the movement immediately.

Explanation

When the movement along an axis must be immediately stopped, this function stops the movement and outputs alarm DS5550 "AXIS IMMEDIATE STOP" by setting the axis immediate stop start signal ESTPR to "1". This function is enabled by setting bit 2 (EST) of parameter No. 1605 to 1.

Acceleration rate at the axis immediate stop

When the movement along an axis must be immediately stopped in the AI contour control mode, this function changes the acceleration rate in look-ahead acceleration/deceleration before interpolation to the value set in parameter No. 1673 or No.1700 (Maximum allowable acceleration rate in the axis immediate stop for each axis), and stops the movement immediately. When the axis immediate stop is executed by the set value of parameter No.1700, the deceleration stop of axis movement is executed by the lowest allowable acceleration rate among the movement axes. When the value set in parameter No. 1673 or selected maximum allowable acceleration rate of parameter No.1700 is smaller than the acceleration rate in look-ahead acceleration/deceleration before interpolation, this function immediately stops the movement at the acceleration rate in look-ahead acceleration/deceleration before interpolation. When the type of acceleration/deceleration is look-ahead bell-shaped acceleration/deceleration before interpolation, this function changes the type to look-ahead linear acceleration/deceleration before interpolation.

When look-ahead acceleration/deceleration before interpolation is used for cutting feed or rapid traverse, this function changes both the acceleration rate and type of acceleration/deceleration.

If 0 or a value lower than the acceleration of look-ahead acceleration/deceleration before interpolation is set to the maximum allowable acceleration rate of parameter No.1673 or No.1700, the acceleration rate and the acceleration/deceleration type of look-ahead acceleration/deceleration before interpolation is not changed, and the tool stops by using the current setting of look-ahead acceleration/deceleration before interpolation.

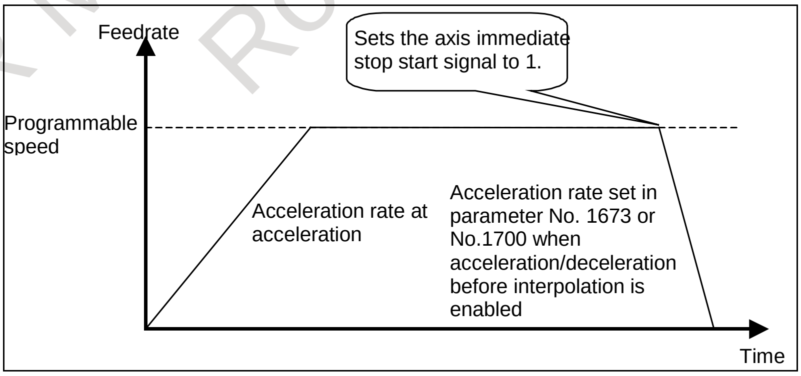


Fig. 1.13 (a)

Acceleration/deceleration after interpolation

Acceleration/deceleration after interpolation is also enabled at the axis immediate stop.

When look-ahead acceleration/deceleration before interpolation is not used, the movement is stopped using the time constant used for acceleration/deceleration after interpolation.

Manual operation

This function is also available during manual operation.

Time chart

The timing chart of the axis immediate stop start signal and deceleration state is shown below (Fig. 1.13 (b)).

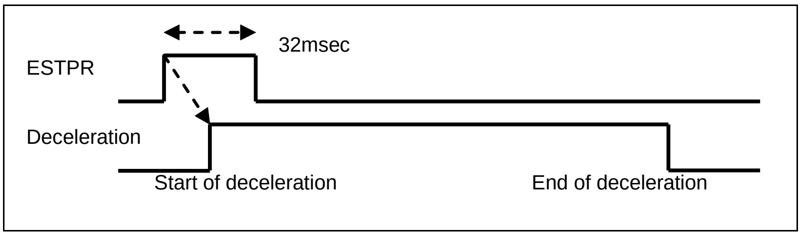


Fig. 1.13 (b)

Arbitrary angular axis control

When the axis immediate stop function is executed for an axis under the arbitrary angular axis control, alarm (PW0000) is issued after the immediate stop.
In this case, the power must be turned off before operation is continued.

Signal

Axis immediate stop start signal ESTPR<Gn203.3>

[Classification] Input signal
[Function] The axis immediate stop function is used to start stopping the movement along an axis.
[Operation] When this signal becomes "1", an axis immediate stop starts.
The width of this signal requires at least 32 msec.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn203					ESTPR			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1605						EST		

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#2 EST Axis immediate stop function is:
0: Disabled.
1: Enabled.

1673	Maximum allowable acceleration rate in tangent direction at axis immediate stop

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/ sec^2, inch/ sec^2, degree/ sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

This parameter sets the maximum allowable acceleration rate in the tangent direction for look-ahead acceleration/deceleration before interpolation at a feed axis immediate stop.
If the parameter is set to a value equal to or greater than 100000.0, the value is clamped to 100000.0.
If 0 or a value lower than the acceleration of look-ahead acceleration/deceleration before interpolation is set, the tool stops by using the current setting without making the following changes:

- Change to the acceleration of look-ahead acceleration/deceleration before interpolation.
- Change to the acceleration/deceleration type from look-ahead bell-shaped acceleration/ deceleration before interpolation to look-ahead linear acceleration/deceleration before interpolation.

1700	Maximum allowable acceleration rate in the axis immediate stop for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

Set a maximum allowable acceleration rate of each axis in look-ahead acceleration/ deceleration before interpolation on the axis immediate stop. The movement axis stops immediately by the lowest allowable acceleration rate in the movement axis. The following changes are executed by axis immediate stop.

- Change the acceleration rate in look-ahead acceleration/deceleration before interpolation.
- Change the look-ahead acceleration/deceleration type from bell-shaped acceleration/deceleration before interpolation to look-ahead linear acceleration/ deceleration before interpolation.

If a value greater than 100000.0 is set, the value is clamped to 100000.0.
If a value lower than the acceleration of look-ahead acceleration/deceleration before interpolation is set, the tool stops by using the current setting of look-ahead acceleration/deceleration before interpolation.

NOTE

If set the value to one or more axis in the parameter No.1700, maximum allowable acceleration rate in the axis immediate stop for each axis becomes effective. And the setting of parameter No.1673 becomes invalid.
In condition the setting of parameter No.1700 is valid, it is considered that the acceleration of look-ahead acceleration/ deceleration before interpolation is set for the axis to which the value is not set.

Alarm and message

Number	Message	Description
DS5550	AXIS IMMEDIATE STOP	The movement along an axis was stopped immediately by the axis immediate stop function.

Notes

NOTE

1 The movement along axes according to the following functions are not stopped with this function:
- PMC axis control
- High precision oscillation function
- Polygon turning
- EGB function
- Live tool control with servo motor

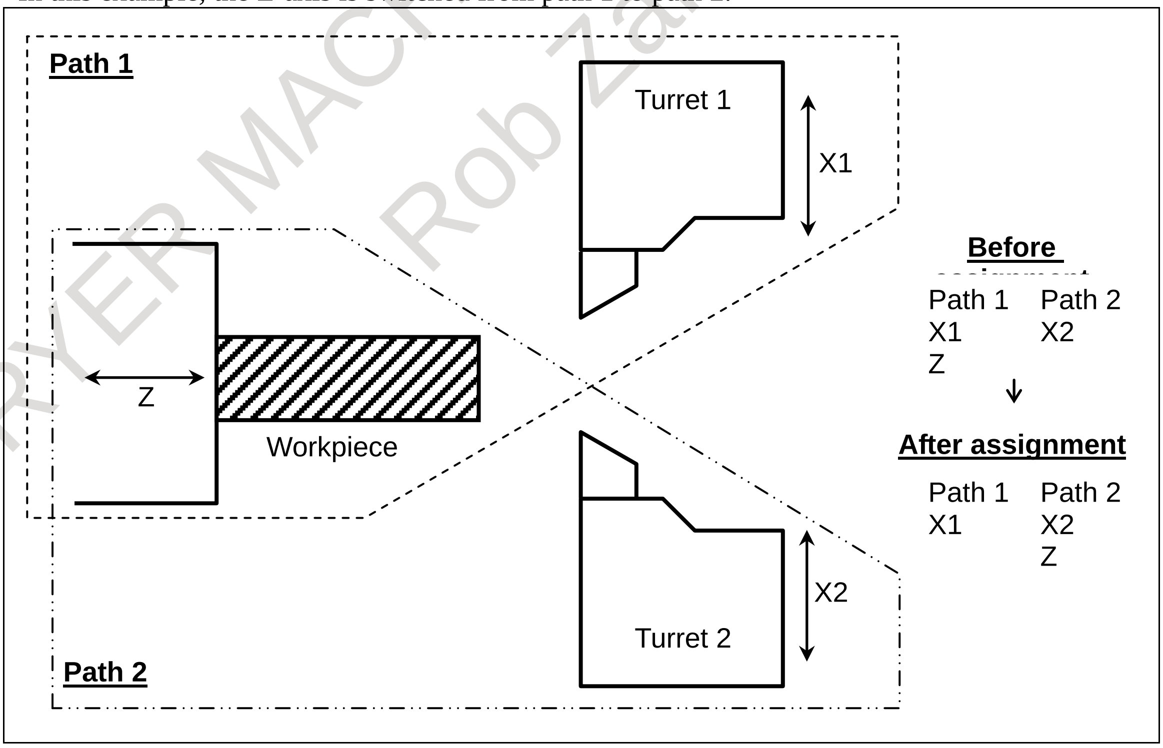
1.14 FLEXIBLE PATH AXIS ASSIGNMENT

Overview

Conventionally, each controlled axis has been controlled within each path. This function can remove each controlled axis from the control of each path and assign them as the controlled axis in the other path.
Using this function makes it possible to control one motor in multiple paths. For example, in the machine having the axis configuration shown in Example 1 (X1 and Z in path 1 and X2 in path 2), the Z-axis can be removed from path 1 and assigned to path 2 to form a different axis configuration (X1 in path 1 and X2 and Z in path 2), therefore requiring no dummy axis unlike composite control.
In the rotary index machine shown in Example 2, axes can be switched among paths.
If an assignment command is issued for an axis yet to be removed, the command waits for the axis to be removed. In this case, no waiting M code is needed.
The new axis configuration (after flexible path axis assignment) is preserved even after the CNC power is turned off.

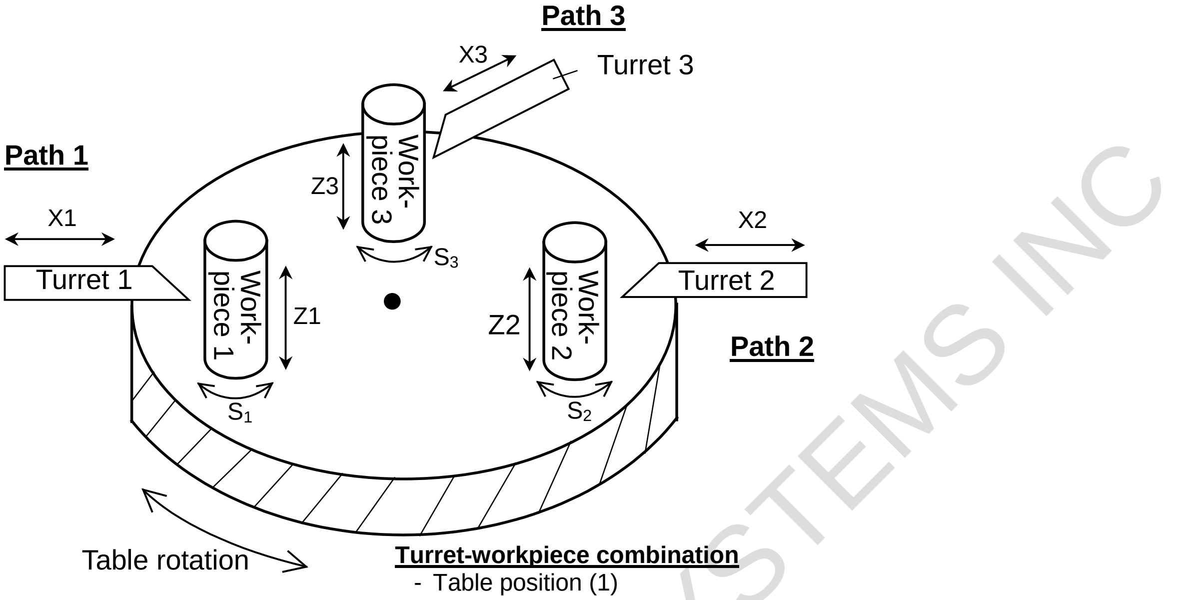
(Example 1)

In this example, the Z-axis is switched from path 1 to path 2.



(Example 2)

In this example, the Z1 axis is switched from path 1 to path 2 or 3.
(Rotary index machine)



Turret-workpiece combination

- Table position (1)
  Axis configuration: Path 1(X1－Z1), Path 2(X2－Z2), Path 3(X3－Z3)
  ↓
- Table position (2)
  Z1, Z2, and Z3 are removed.
  Z2, Z3, and Z1 are assigned, respectively, to paths 1, 2, and 3.
  Axis configuration: Path 1(X1－Z2), Path 2(X2－Z3), Path 3(X3－Z1)
  ↓
- Table position (3)
  Z1, Z2, and Z3 are removed.
  Z3, Z1, and Z2 are assigned, respectively, to paths 1, 2, and 3.
  Axis configuration: Path 1(X1－Z3), Path 2(X2－Z1), Path 3(X3－Z2)

The flexible path axis assignment function provides the following three commands.

1. Controlled-axis removal command
   A specified axis is removed from under control of a specified path.
   No CNC program can direct the removed axis any more.
2. Controlled-axis assignment command
   A specified axis is placed under control of a specified path.
3. Controlled-axis exchange command
   Two specified axes can be exchanged directly.

Explanation

- ID number- and name-based axis assignment methods

- ID number type (bit 3 (FAM) of parameter No.11561 is set to 0)
  The term ID number refers to a unique identification number (parameter No. 11560) for distinguishing a specific axis.
  The ID number-based method can use the ID numbers for axes after address P, Q, or R in the G code stated below to reassign the axes.
  This type of axis assignment can be used even on a machine having two or more axes having the same name.
- Axis name type (bit 3 (FAM) of parameter No.11561 is set to 1)
  Unlike the ID number-based method, this method can use the names of axes in the G to reassign them.
  The method can be used on a machine whose axes have fixed, unique names.

- Format

1) ID number type (bit 3 (FAM) of parameter No.11561 is set to 0)

G52.1 P_ Q_ R_ ; **Command to remove axes**
(Issued to a path having axes to be removed)

P,Q,R : ID numbers for axes to be removed (up to 3 axes can be removed at a time)

G52.2 P_ Q_ R_ I_ J_ K_ ; **Command to assign axes**
(Issued to a path to which axes are to be assigned)

P,Q,R : ID numbers for axes to be assigned (up to 3 axes can be assigned at a time)
I,J,K : Axis order numbers where axes are to be inserted
(An axis position in a path is specified. "I and P","J and Q", and "K and R" are paired. If an axis order number for an axis is omitted, the axis is assigned as the last axis in the path.)

G52.3 P_ Q_ (L_) ; **Command to exchange axes between paths**
(Issued to paths between which axes are to be exchanged)

P : ID number of an axis to be exchanged (in a source path)
Q : ID number of an axis to be exchanged (in a destination path)
L : Destination path number (omissible in a 2-path system)

ID numbers specified in this parameter (No. 11560) must be unique values that can distinguish each axis from one another.

2) Axis name type (bit 3 (FAM) of parameter No.11561 is set to 1)

G52.1 IP0 ; **Command to remove axes**
(Issued to a path having axes to be removed)

IP : ID numbers for axes to be removed (up to 3 axes can be removed at a time)

G52.2 IP_ ; **Command to assign axes**
(Issued to a path to which axes are to be assigned)

IP_ : Names of axes to be assigned (up to 3 axes can be assigned at a time) and axis order numbers (specifying where to assign each axis in a path)

G52.3 IP0 (L_) ; **Command to exchange axes having the same name between paths**

G52.3 IP$_1$0 IP$_2$0 (L_) ; **Command to exchange axes having different names between paths**
(Issued to paths between which axes are to be exchanged)

IP$_1$: Axis name of an axis to be exchanged (in a source path)
IP$_2$: Axis name of an axis to be exchanged (in a destination path)
L : Destination path number (omissible in a 2-path system)

NOTE

1 Be sure to specify ID numbers (parameter No. 11560) even when using the axis name-based assignment method.
2 G52.1, G52.2, and G52.3 are a one-shot G code in group 00.
3 Do not use G52.1, G52.2, and G52.3 together in a single block; they must be used in separate blocks.
4 When axis exchange command G52.3 is issued (in a certain path) for the first time, the axes of interest are put in a wait state. Actual axis exchange takes place next time the command is issued (to the other path).
5 Alarm PS0514 "ILLEGAL COMMAND IN FLEXIBLE PATH AXIS ASSIGNMENT" is issued if a conflict occurs between exchange command G52.3 axis specifications.
6 Alarm PS0514 is issued if an attempt is made to exchange (G52.3) axes within a path.
7 When axis removal, assignment, or exchange is performed actually, be sure to keep the axes of interest at halt.
8 Do not use a decimal point in ID numbers, axis order numbers, or path numbers specified in G52.1, G52.2, or G52.3.

(Example)	Enabled G52.2 P1 I1	Disabled G52.2 P1.0 I1.0
	Enabled G52.2 C1	Disabled G52.2 C1.0

- Specification of the order of axes to be assigned

If a program contains an invalid axis order number specification, the CNC behaves as follows:
Example)

Axis configuration: X,Z
Setting value of Parameter No. 11560: (X,Y,Z,C) = (1, 2, 3, 4)

1) If no axis order is specified:
   Command: G52.2 P2 Q4 ;
   The Y- and C-axes are assigned, respectively, as the third and fourth (last) axes. (X,Z,Y,C)
2) If there is a duplicate axis order specification:
   Command: G52.2 P2 I2 Q4 J2 ;
   Because "I2" and "J2" are doubly specified as the second axis, alarm PS0514 is issued.
3) If a command does not match the axis configuration:
   Command: G52.2 P2 I2 Q4 J5 ;
   Because there is a conflict with "J5" specified as the fifth axis, alarm PS0514 is issued.

- Controlled-axis removal and assignment

- Removal

If an axis is specified in axis removal command G52.1, it is removed from its path. No command can be executed to the removed command any more. Alarm PS0009 "IMPROPER NC-ADDRESS" is issued for any command directed to the removed axis.

On the position screen, the following axis status 'R' is displayed for the removed axis.

- Assignment

An arbitrary axis can be assigned using assignment axis command G52.2. Its arguments (with the ID number-based method, I, J, and K and, with the axis name-based method, numbers that follow axis names) can be used to place axes in arbitrary positions in the path of interest.

If a specified axis is yet to be removed, bit 1 (FAW) of parameter No. 11561 can be used to specify whether to defer the command execution until after the axis is removed or to issue alarm PS0514.

When the removed axis is assigned again, its status display 'R' disappears from the position screen.

Program example

1) ID number type (bit 3 (FAM) of parameter No.11561 is set to 0)
Axis configuration and setting of parameter No.11560

Path 1		Path 2	
X	0	X	0
Z	0	Z	0
C	103	A	0
A	0		

1. Removal and assignment commands (with no alarm)

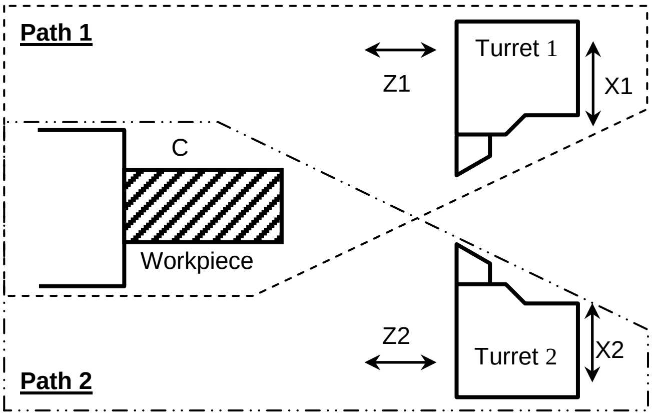
Program		Operation		
Path 1	**Path 2**		**Path 1**	**Path 2**
N1 G00 X1.0 ;	N1 G00 Z1.0 ;	Operation	X move	Z move
		Axis configuration	X,Z,C,A	X,Z,A
N2 **G52.1 P103** ;	N2 G00 Z2.0 ;	Operation	C is removed.	Z move
		Axis configuration	**X**,**Z**,**A**	X,Z,A
N3 G00 X2.0 ;	N3 **G52.2 P103 I3** ;	Operation	X move	The C-axis is assigned as the third axis.
		Axis configuration	X,Z,A	X,Z,**'C'**,A

2. Commands getting in a wait state (bit 1 (FAW) of parameter No. 11561 = 0)

Program		Operation		
Path 1	**Path 2**		**Path 1**	**Path 2**
N1 G00 X1.0 ;	N1 G00 Z1.0 ;	Operation	X move	Z move
		Axis configuration	X,Z,C,A	X,Z,A
N2 G00 X2.0 ;	N2 **G52.2 P103 I3** ;	Operation	X move	Waiting for C to be removed
		Axis configuration	X,Z,C,A	X,Z,A
N3 **G52.1 P103** ;		Operation	C is removed.	After C is removed from path 1, it is assigned as the third axis in path 2.
		Axis configuration	**X**,**Z**,**A**	X,Z,**'C'**,A
N4 G00 X3.0 ;	N3 G00 Z2.0 ;	Operation	X move	Z move
		Axis configuration	X,Z,A	X,Z,C,A

3. Commands leading to alarm PS0514 (bit 1 (FAW) of parameter No. 11561 = 1)

Program		Operation		
Path 1	**Path 2**		**Path 1**	**Path 2**
N1 G00 X1.0 ;	N1 G00 Z1.0 ;	Operation	X move	Z move
		Axis configuration	X,Z,C,A	X,Z,A
N2 G00 X2.0 ;	N2 **G52.2 P103 I3** ;	Operation	X move	Alarm PS0514 is issued because an attempt was made to assign C, yet to be removed, as the third axis in path 2.
		Axis configuration	X,Z,C,A	X,Z,A

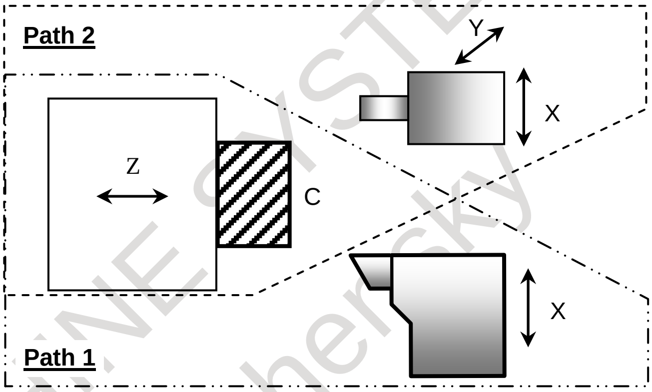


2) Axis name type (bit 3 (FAM) of parameter No.11561 is set to 1)

a) Invalid extended axis name (bit 0 (EEA) of parameter No.1000 is set to 0)

Axis configuration

Path 1	Path 2
X	X
Z	Y
C	-



Program		Operation		
Path 1	Path 2		Path 1	Path 2
N1 G00 X1.0 ;	N1 G00 Y1.0 ;	Operation	X move	Y move
		Axis configuration	X,Z,C	X,Y
N2 **G52.1 Z0 C0** ;	N2 G00 Y2.0 ;	Operation	Z and C are removed.	Y move
		Axis configuration	**X**	X,Y
N3 G00 X2.0 ;	N3 **G52.2 Z3 C4** ;	Operation	X move	Z and C from path 1 are assigned, respectively, as the third and fourth axes.
		Axis configuration	X	X,Y,**'Z'**,**'C'**

b) Valid extended axis name (bit 0 (EEA) of parameter No.1000 is set to 1)

Axis configuration

Path 1	Path 2
XA	XB
ZA	YB
CA	-

Program		Operation		
Path 1	Path 2		Path 1	Path 2
N1 G00 XA=1.0 ;	N1 G00 YB=1.0 ;	Operation	XA move	YB move
		Axis configuration	XA,ZA,CA	XB,YB
N2 **G52.1 ZA=0 CA=0** ;	N2 G00 YB=2.0 ;	Operation	ZA and CA are removed.	YB move
		Axis configuration	**XA**	XB,YB
N3 G00 XA=2.0 ;	N3 **G52.2 ZA=3 CA=4** ;	Operation	XA move	ZA and CA from path1 are assigned, respectively, as the third and fourth axes.
		Axis configuration	XA	XB,YB,**'ZA'**,**'CA'**

Controlled-axis exchange

Axis exchange command G52.3 can be used to exchange arbitrary axes with each other.
Axis names used after exchange are determined according to the setting of bit 1 (FAN) of parameter No. 11562 as follows:

- If parameter FAN = 0, the name previously given to each axis is inherited.
- If parameter FAN = 1, the axis names are also exchanged.

If the setting of the parameter FAN conflicts between paths, FAN = 0 is assumed.

> **NOTE**
> When the setting of bit 1 (FAN) of parameter No. 11562 is valid, an attempt to exchange an axis using an extended axis name with one using no extended axis name resulting in no axis name being displayed normally. Never attempt axis exchange between an axis using an extended axis name and one using no extended axis name.

- Program example
  1) ID number type (bit 3 (FAM) of parameter No.11561 is set to 0)
     Axis configuration and setting of parameter No.11560

Path 1		Path 2		Path 3	
X	0	X	0	X	0
Z	102	Y	202	Z	302
A	103	B	203	C	303

1. Exchange command (with no alarm)

Program			Operation			
Path 1	**Path 2**	**Path 3**		**Path 1**	**Path 2**	**Path 3**
N1 G00 X1.0 ;	N1 G00 Y1.0 ;	N1 G00 X1.0 ;	Operation	X move	Y move	X move
			Axis configuration	X,Z,A	X,Y,B	X,Z,C
N2 **G52.3 P103 Q203 L2** ;	N2 **G52.3 P203 Q103 L1** ;	N2 **G52.3 P303 Q103 L2** ;	Operation	A from path 1 is exchanged with B from path 2.	B from path 2 is exchanged with A from path 1.	The CNC waits for block N3 in path 2.
			Axis configuration	X,Z,**'B'**	X,Y,**'A'**	X,Z,C
N3 G00 B90.0;	N3 **G52.3 P103 Q303 L3** ;		Operation	B move	A from path 2 is exchanged with C from path 3.	After execution of block N3 in path 2, C from path 3 is exchanged with A from path 2.
			Axis configuration	X,Z,B	X,Y,**'C'**	X,Z,**'A'**

2. When alarm PS0514 is issued

Program			Operation			
Path 1	**Path 2**	**Path 3**		**Path 1**	**Path 2**	**Path 3**
N1 G00 X1.0 ;	N1 G00 Y1.0 ;	N1 G00 X1.0 ;	Operation	X move	Y move	X move
			Axis configuration	X,Z,A	X,Y,B	X,Z,C
N2 **G52.3 P103 Q203 L2** ;	N2 **G52.3 P203 Q102 L1** ;	N2 G00 X2.0 ;	Operation	Alarm PS0514 is issued because of an axis ID number specification conflict between exchange commands. (*1)	Alarm PS0514 is issued because of an axis ID number specification conflict between exchange commands. (*1)	X move
			Axis configuration	X,Z,A	X,Y,B	X,Z,C

*1: The alarm is issued on the path to which an exchange command is issued later.

3. When exchange commands are used in more than two paths (between paths 1 and 2 and between paths 2 and 3)

Program			Operation			
Path 1	**Path 2**	**Path 3**		**Path 1**	**Path 2**	**Path 3**
N1 G00 X1.0 ;	N1 G00 Y1.0 ;	N1 G00 X1.0 ;	Operation	X move	Y move	X move
			Axis configuration	X,Z,A	X,Y,B	X,Z,C
N2 **G52.3 P103 Q203 L2** ;	N2 **G52.3 P202 Q302 L3** ;	N2 G00 X2.0 ;	Operation	The CNC waits for block N3 in path 2.	The CNC waits for block N4 in path 3.	X move
			Axis configuration	X,Z,A	X,Y,B	X,Z,C
		N3 G00 X3.0 ;	Operation	The CNC waits for block N3 in path 2.	The CNC waits for block N4 in path 3.	X move
			Axis configuration	X,Z,A	X,Y,B	X,Z,C
		N4 **G52.3 P302 Q202 L2** ;	Operation	The CNC waits for block N3 in path 2.	After execution of block N4 in path 3, Y from path 2 is exchanged with Z from path 3.	Z from path 3 is exchanged with Y from path 2.
			Axis configuration	X,Z,A	X,**'Z'**,B	X,**'Y'**,C
	N3 **G52.3 P203 Q103 L1** ;	N5 G00 X4.0 ;	Operation	After execution of block N3 in path 2, A from path 1 is exchanged with B from path 2.	B from path 2 is exchanged with A from path 1.	X move
			Axis configuration	X,Z,**'B'**	X,Z,**'A'**	X,Y,C
N3 G00 X2.0 ;	N4 G00 Z2.0 ;	N6 G00 Y5.0 ;	Operation	X move	Z move	Y move
			Axis configuration	X,Z,B	X,Z,A	X,Y,C

2) Axis name type (bit 3 (FAM) of parameter No.11561 is set to 1)

a) Invalid extended axis name (bit 0 (EEA) of parameter No.1000 is set to 0)
Axis configuration

Path 1	Path 2	Path 3
X	X	X
Z	Y	Z
A	B	C

1. Exchanging axes having the same name

Program			Operation			
Path 1	Path 2	Path 3		Path 1	Path 2	Path 3
N1 G00 X1.0 ;	N1 G00 X1.0 ;	N1 G00 X1.0 ;	Operation	X move	X move	X move
			Axis configuration	X,Z,A	X,Y,B	X,Z,C
N2 **G52.3 X0 L2** ;	N2 **G52.3 X0 L1** ;	N2 **G52.3 Z0 L1** ;	Operation	X from path 1 is exchanged X from path 2.	X from path 2 is exchanged X from path 1.	The CNC waits for block N3 in path 1.
			Axis configuration	**'X'**,Z,A	**'X'**,Y,B	X,Z,C
N3 **G52.3 Z0 L3** ;	N3 G00 X0.0 ;		Operation	Z from path 1 is exchanged Z from path 3.	X move	After execution of block N3 in path 1, Z from path 1 is exchanged with Z from path 3.
			Axis configuration	X,**'Z'**,A	X,Y,B	X,**'Z'**,C

2. Exchanging axes having different same names

Program			Operation			
Path 1	Path 2	Path 3		Path 1	Path 2	Path 3
N1 G00 X1.0 ;	N1 G00 X1.0 ;	N1 G00 X1.0 ;	Operation	X move	X move	X move
			Axis configuration	X,Z,A	X,Y,B	X,Z,C
N2 **G52.3 A0 B0 L2** ;	N2 **G52.3 B0 A0 L1** ;	N2 **G52.3 C0 B0 L1** ;	Operation	A from path 1 is exchanged with B from path 2.	B from path 2 is exchanged with A from path 1.	The CNC waits for block N3 in path 1.
			Axis configuration	X,Z,**'B'**	X,Y,**'A'**	X,Z,C
N3 **G52.3 B0 C0 L3** ;	N3 G00 A180.0;		Operation	B from path 1 is exchanged with C from path 3.	A move	After execution of block N3 in path 1, C from path 3 is exchanged with B from path 1.
			Axis configuration	X,Z,**'C'**	X,Y,A	X,Z,**'B'**

b) Valid extended axis name (bit 0 (EEA) of parameter No.1000 is set to 1)
Axis configuration

Path 1	Path 2	Path 3
XA	XB	XC
ZA	ZB	ZC
CA	AB	CC
AA		

Program			Operation			
Path 1	**Path 2**	**Path 3**		**Path 1**	**Path 2**	**Path 3**
N1 G00 XA=1.0 ;	N1 G00 ZB=1.0 ;	N1 G00 XC=1.0 ;	Operation	XA move	ZB move	XC move
			Axis configuration	XA,ZA,CA,AA	XB,ZB,AB	XC,ZC,CC
N2 **G52.3 AA=0 AB=0 L2** ;	N2 **G52.3 AB=0 AA=0 L1** ;	N2 **G52.3 CC=0 AA=0 L2** ;	Operation	AA from path 1 is exchanged with AB from path 2.	AB from path 2 is exchanged with AA from path 1.	The CNC waits for block N3 in path 2.
			Axis configuration	XA,ZA,CA,**'AB'**	XB,ZB,**'AA'**	XC,ZC,CC
N3 G00 AB=5.0;	N3 **G52.3 AA=0 CC=0 L3** ;		Operation	AB move	AA from path 2 is exchanged with CC from path 3.	After execution of block N3 in path 2, CC from path 3 is exchanged with AA from path 2.
			Axis configuration	XA,ZA,CA,AB	XB,ZB,**'CC'**	XC,ZC,**'AA'**

- Fixed axis name of each path

The name of the axis, which assigned by flexible path axis assignment, can be changed to the name which specified in each path.
Set the axis name of assigned axis to parameter No.11555, and axis suffix to parameter No.11556 and No.11557. Select the axis which changes the name in the each path by the bit 0 of parameter No. 1l564(PAN). If the selected axis is assigned to another path by flexible path axis assignment, the axis name is changed to the name specified in the each path.
This setting is available on only one axis in a path.

Example) Identification number (parameter No.11560)(Z11,Z21,Z31)=(102,202,302),
Specified axis name (No.11555) = 49(path1), 49(path2), 49(path3),
Subscript (No.11556)=49(path1), 50(path2), 51(path3),
No.11557 = 49(path1), 49(path2) 49(path3),
Bit 0(PAN) of parameter No.11564=1(path1), 1(path2), 1(path3),
Bit 3(FAM) of parameter No.11561 (FAM)=1

Example of the program for each path

Block number	Path 1	Path 2	Path 3	
	O1001	O2001	O3001	
N0010	G52.1 Z11=0 ;	G52.1 Z21=0 ;	G52.1 Z31=0 ;	O1001: Release Z11(ID 102) O2001: Release Z21(ID 202) O3001: Release Z31(ID 302)
N0100	G52.2 Z21=2 ;	G52.2 Z31=2 ;	G52.2 Z11=2 ;	O1001: Assign Z21(ID 202) O2001: Assign Z31(ID 302) O3001: Assign Z11(ID 102)
:	:	:	:	:
N0020	G52.1 Z11=0 ;	G52.1 Z21=0 ;	G52.1 Z31=0 ;	O1001: Release Z11(ID 202) O2001: Release Z21(ID 302) O3001: ReleaseZ31(ID 102)
N0200	G52.2 Z21=2 ;	G52.2 Z31=2 ;	G52.2 Z11=2 ;	O1001: Assign Z21(ID 302) O2001: Assign Z31(ID 102) O3001: Assign Z11(ID 202)

Block number	Path 1 O1001	Path 2 O2001	Path 3 O3001	
:	:	:	:	:
N0030	G52.1 Z11=0 ;	G52.1 Z21=0 ;	G52.1 Z31=0 ;	O1001: Release Z11(ID 302) O2001: Release Z21(ID 102) O3001: Release Z31(ID 202
N0300	G52.2 Z21=2 ;	G52.2 Z31=2 ;	G52.2 Z11=2 ;	O1001: Assign Z21(ID 102) O2001: Assign Z31(ID 202) O3001: Z11(ID 302)

- The combination of the axis name of each path after axis assignment and identification number of flexible path axis assignment.

Block No.		Path 1			Path 2			Path 3		
N0100	Axis name	X11	Z11	Z12	X21	Z21	Z22	X31	Z31	Z32
	ID number		202			302			102	
N0200	Axis name	X11	Z11	Z12	X21	Z21	Z22	X31	Z31	Z32
	ID number		302			102			202	
N0300	Axis name	X11	Z11	Z12	X21	Z21	Z22	X31	Z31	Z32
	ID number		102			202			302	

- Parameter setting for axis exchange used together with other functions

Some NC functions require specifying relative axis numbers within a path (intra-path relative axis number) using parameters (Tables 1.14(a) to (e)). When axes are exchanged between paths, it is likely that an axis number used with a function in the tables may fail to match a parameter value set as an intra-path relative axis number (Example 1).

Example 1)

In this example, a relative axis number is changed after axis exchange.
(Before axis exchange, C in path 1 and Z in path 2 are, respectively, the fourth and second axes (relative axis number). After axis exchange, C in path 1 and Z in path 2 are the third axes in the respective paths.)

Before assignment			After replacement	
Path 1	Path 2		Path 1	Path 2
X	X	→	X	X
Z	Z		Z	'Y'
'Y'			C	Z
C				

(1) Case in which parameter setting requires using G10

When using functions and parameters listed in Table 1.14 (a) to (c) in such a way that the axis order is changed during axis exchange, re-set the related parameters, using programmable parameter input (G10).

Table 1.14 (a) Functions requiring setting intra-path relative numbers using parameters

Function	Related parameter number
Cs contouring control	3900, 3910, 3920, 3930, 3940
Constant surface speed control	3770
Y-axis offset	5043 to 5045
Canned cycle for grinding	5176 to 5183
Polar coordinate interpolation	5460, 5461
Normal direction control	5480
Flexible synchronization control	5660 to 5667
Position switch	6910 to 6925, 8570 to 8579
High precision oscillation function	8370
5-axis machining function	19657, 19681, 19686

Table 1.14 (b) Functions requiring setting intra-path relative axis numbers for individual axes

Function	Related parameter number
Synchronous control	8180
Composite control	8183
Superimposed Control	8186

Table 1.14 (c) Others

Related parameter number	Description
1020, 1025, 1026	Set axis names.
1022	Set 3 basic parallel axes. (Example: Cylindrical interpolation)

(2) Case in which G10 is unusable for parameter setting

Using functions listed in Tables 1.14 (d) and 1.14 (e) in such a way that the axis order is changed in axis exchange does not allow G10 to be used because the parameters used require turning the power off and on again. Follow the examples given in (3), "Case in which it is unnecessary to use G10 for parameter setting", below.

Table 1.14 (d) Functions requiring setting intra-path relative numbers using parameters (with power to be turned off and on again)

Function	Related parameter number
Y-axis offset	5044, 5045
Index table indexing	5510
Straightness compensation	5711 to 5716 5721 to 5726
Electronic gear box	7710
Arbitrary angular axis control	8211, 8212

Table 1.14 (e) Others (with power to be turned off and on again)

Function	Related parameter number	Note
Tandem control	1817#6	Must be specified for both the master and slave axes.

(3) Case in which it is unnecessary to use G10 for parameter setting

Axis configurations shown in Examples 2 to 4 do not cause intra-path relative numbers to be changed in flexible path axis assignment. So, it is unnecessary to use programmable parameter input (G10) for parameter re-setting.

Example 2)

In this example, axes having the same intra-path relative number are exchanged with each other.

Path 1		Path 2
'X'	↔	'X'
Y		Y
Z		Z
C		C

Example 3)

In this example, an axis is removed or assigned as the last one in the respective paths.
(Polygon turning is performed using the rotary axis “C” as the fifth axis in path 1 and the fourth axis in path 2.)

Before assignment		→	After assignment	
Path 1	Path 2		Path 1	Path 2
X	X		X	X
Y	Y	→	Y	Y
Z	Z		Z	Z
A			A	**'C'**
'C'				

Example 4)

When the display order of axes is set by parameter No.3130 and releasing and getting axis excluding the last display order in the path.
In this case, set the axis as the last order in the path, and set the display order in position screen by No.3130.
(Example of Constant surface speed control : "X" of first display axis in path 1, "X" of first display axis in path 2)

Setting				Axis name on position screen	
Path 1		Path 2			
Physical order	Parameter No.3130	Physical order	Parameter No.3130	Path 1	Path 2
Y	2	Y	2	**'X'**	
Z	3	Z	3	Y	Y
A	4			Z	Z
C	5			A	
'X'	1			C	

↓ After assignment

Physical order	Parameter No.3130	Physical order	Parameter No.3130	Path 1	Path 2
Y	2	Y	2		**'X'**
Z	3	Z	3	Y	Y
A	4	**'X'**	1	Z	Z
C	5			A	
				C	

- Signal specifications

Even after the arrangement of axes has been changed in flexible path axis assignment, the signals involved remain enabled if they are of axis type.

- Reset / Emergency stop
If a reset or emergency stop occurs during flexible path axis assignment, the latest axis configuration is preserved.

- Power-off/-on
If the power is turned off and on again during flexible path axis assignment, the axis configuration is changed or preserved depending on the setting of bit 2 (FA0) of parameter No. 11561 as follows:
- If parameter FA0 = 0, the initial axis configuration (parameter No. 0981) is resumed.
- If parameter FA0 = 1, the latest axis configuration is preserved.

- Number of controlled paths in a path
Axis assignment cannot exceed the maximum allowable number of controlled axes in each path. The number includes any removed axes. No removal command can make the number of controllable axes in each path become zero. Such an attempt leads to alarm PS0514.

- Offset
The various offsets such as the cutter compensation and tool length compensation must be kept canceled during execution of flexible path axis assignment. Otherwise, alarm PS0514 is issued.

- Workpiece Coordinate System
Once flexible path axis assignment is executed, it is necessary to select the workpiece coordinate system again.

- Diameter / Radius setting
When arbitrary flexible path axis assignment is executed, diameter or radius setting is in accordance with the parameter DIA (No. 1006 # 3).

- Plane selection
When Flexible Path Axis Assignment is executed to the axis which consist the current plane selection, select the plane again after the Flexible Path Axis Assignment is finished.

- Cs contour controlled axis
To perform Cs contour control, use the Cs contour control change signals CONS1 to CONS4 <Gn274.0 to 3> for each spindle.
No Cs contour control mode changeover can be made for any removed axis.
Gain switching is carried out as follows:
Removing the Cs contour control axis: The gain specified with parameter No. 1825 is resumed.
Assigning the Cs contour control axis: Switching occurs to the gain specified using parameters Nos. 3900 to 3944 for the path to which the Cs contour control axis belongs.
Exchanging Cs contour control axes: Switching occurs to the gain specified using Nos. 3900 to 3944 for the path to which the respective Cs contour control axes belong.

- Reference axis
The reference axis is decided according to the arrangement of axes valid after flexible path axis assignment.

- State of waiting for removal
Alarm PS0514 is issued if either of the following commands is issued from the other path when the CNC is waiting for an axis to be removed.
- Axis exchange command issued from the other path.
- Axis assignment command issued from the other path to an axis yet to be removed

- Dwell

When issuing a dwell command, use address P. If flexible path axis assignment results in no X-axis existing in the path of interest or causes the reference axis to be renamed, it is impossible for the dwell command to use address X or the name of the reference axis.
Alarm PS0009 may be issued if the dwell command attempts to use address X or the reference axis name.

- Axis synchronous control

Alarm PS0514 is issued if an attempt is made to perform flexible path axis assignment for an axis under axis synchronous control. Before performing flexible path axis assignment for an axis, cancel axis synchronous control for that axis.
When flexible path axis assignment changes the arrangement of axes not under axis synchronous control:

- Even if flexible path axis assignment changes the arrangement of the master axis, the master axis is preserved.
- Even if flexible path axis assignment changes the arrangement of axes during axis synchronous control, the relationship between the master and slave axes is preserved.

- Constant surface speed control

If flexible path axis assignment changed the arrangement of axes, which axis to use as a reference in constant-surface speed control calculation is determined according to the axis arrangement used when a constant-surface speed control command (G96) is issued.
Even if the arrangement of axes is changed during constant-surface speed control, the calculation reference axis is preserved until the constant-surface speed control is canceled.

- PMC axis control

No flexible path axis assignment can be performed for any axis under PMC axis control.
Before performing flexible path axis assignment, cancel PMC axis control.
No PMC axis control can be performed for any removed axis. Before performing PMC axis control for an axis, assign that axis.

- Synchronous/Composite control

Alarm PS0514 is issued if an attempt is made to perform flexible path axis assignment for axes under synchronous control or composite control. Before performing flexible path axis assignment, cancel synchronous control and composite control.
If synchronous control or composite control is performed with the arrangement of axes changed in flexible path axis assignment, synchronous control or composite control takes effect on the new axis arrangement.

- Superimposed control

Alarm PS0514 is issued if an attempt is made to perform flexible path axis arrangement for axes under superimposed control. Before performing flexible path axis arrangement, cancel superimposed control.
If superimposed control is performed with the arrangement of axes changed in flexible path axis assignment, superimposed control takes effect on the new axis arrangement.

- Spindle control with servo motor

If flexible path axis assignment is performed for axes under Spindle control with servo motor:

- Alarm PS0514 is issued if flexible path axis assignment is performed during the SV speed control mode. Before performing flexible path axis assignment, switch to the positional control mode.
- If it is necessary to switch to the SV speed control mode for Spindle control with servo motor, do so on the path to which the axis under Spindle control with servo motor belongs.
- Issuing a command for servo motor-based rigid tapping by changing the arrangement of axes under Spindle control with servo motor requires setting rigid tapping based on the spindle of the other spindle.
- No SV speed control mode switching command can be issued for any removed axis. Issuing such a command results in alarm PS0514.

- Parameter No. 3130 (axis display order on the current position display screen)

When using parameter No. 3130, take the following into consideration.

- The parameter settings for an axis newly assigned by flexible path axis assignment are valid for the path to which the new axis is assigned. When specifying parameter No. 3130 or rewriting it using G10, take the conditions of the axis to be assigned into consideration.
- If parameter No. 3130 is already specified, the current position is displayed according to the setting of the parameter No. 3130 even when flexible path axis assignment is performed to change the arrangement of axes.
- When the axis to which parameter No. 3130 is set is removed, the current position display of the removed axis is displayed according to the setting of the parameter No. 3130. When the axis to which parameter No. 3130 is set is assigned, the current position display may include blanks.

- Polygon turning

Even when the axis configuration or axis arrangement is changed by flexible path axis assignment, an axis can be recognized as a tool rotation axis if the axis belongs to a path where a polygon turning command is issued.

If a path where a polygon turning command is issued has no tool rotation axis specified with a parameter when the power is turned on, the tool rotation axis is determined according to the axis arrangement that is valid when a polygon turning command is issued.

When re-assigning a tool rotation axis for polygon turning by flexible path axis assignment, it is necessary to take the axis arrangement into consideration.

- Inch/metric input

No flexible path axis assignment can be applied between paths using different input unit methods (inch input and metric input).

- Extended external machine zero point shift

When apply the Extended external machine zero point shift to the axis which the index in the path has changed by Flexible Path Axis Assignment, set the parameter FES(bit 4 of parameter No.11565) to 1. When the parameter FES is 1, the R addresses of each axis which specifies the shift value of Extended external machine zero point shift is held to the initial order of axis, and becomes not to change by Flexible path axis assignment.

- Position switch

When apply the Position switch to the axis which the index in the path has changed by Flexible Path Axis Assignment, set the parameter FPS(bit 5 of parameter No.11565) to 1. When the parameter FPS is 1, the target axis of each position switch is held to the initial order of axis, and becomes not to change by Flexible path axis assignment.

- 3-dimensional coordinate conversion

In 3-dimensional coordinate conversion mode, Flexible Path Axis Assignment cannot be performed to the target axis of the conversion.

- Number of blocks read ahead

An alarm (PS0514 or DS0080) will issued if an arbitrary axis switching command that satisfies all of the following conditions is performed.

- Number of blocks read ahead differs between paths.
- The total of the axis belonging to the path with the largest number of blocks read ahead at startup and the axis belonging to the other path and being the object of Flexible Path Axis Assignment (the axis whose parameter No. 11560 is not 0) exceeds the maximum for allowable number of controlled axes in each path.
- At startup, the axes that belonged to a path with a small number of blocks read ahead were moved to a path with a large number of blocks read ahead.

In the above case, if the axis on which flexible path axis assignment is performed to a path with a large number of blocks read ahead is limited to a part of the axes for which parameter No. 11560 is set in the other path, bit 3(FHM) of parameter No.11565 is set to 1, disable the automatic judgment of the axis that extends the number of blocks read ahead, then bit 2(FHE) of parameter No.11562 is set to 1 for those axes.
However, the number of axes that can be set to 1 of FHE is up to the system maximum number of control axes minus the number of control axes of the system with the largest number of blocks read ahead.

Diagnosis data

4000	Reason number of alarm in flexible path axis assignment

The cause of the alarm that may be issued in flexible path axis assignment is displayed.

1 The number of axes in the path is 0.
2 The number of axes in the path is larger than its allowable maximum value.
3 The removal command has no ID specification.
4 The removal command has a duplicate ID specification.
5 An axis specified with removal command P does not exist in the path or has been removed from the path.
6 An axis specified with removal command Q does not exist in the path or has been removed from the path.
7 An axis specified with removal command R does not exist in the path or has been removed from the path.
8 An axis specified with the removal command does not exist in the path or has been removed from the path.
9 The removal command has no axis specification or has an ID specification.
10 In flexible path axis assignment, the ID specification is incorrect.
11 The assignment command has no ID specification.
12 The assignment command has a duplicate ID specification.
13 The assignment command has a duplicate axis arrangement specification.
14 The path specified with the arrangement command has no target axis or the arrangement command has no ID specification.
15 The path specified with the arrangement command has an invalid axis assignment specification.
16 An axis whose removal a command is waiting for belongs to the path where the command was issued.
18 An axis whose removal an exchange command is waiting for belongs to the path paired with the path where the exchange command was issued.
19 An axis for which an assignment command was issued is yet to be removed. (Bit 1 of parameter No. 11561 is set to 1.)
20 An axis for which an assignment command is issued in a path belongs to another path where a removal command for it has been issued.
21 An axis for which an assignment command was issued is yet to be removed.
22 The assignment command has no axis specification or has an ID specification.
23 G-code of axis removal, axis assignment, or axis exchange of Flexible Path Axis Assignment is executed when the signal type of Flexible Path Axis Assignment is selected (bit 0 (CSG) of parameter No.11563 is 1).
24 An axis at which an exchange command is targeted belongs to the path where the exchange command was issued.
25 The exchange command has no ID specification.
26 The exchange command has a duplicate ID specification.
27 In a system having 3 or more paths, an exchange command has no L specification.
28 An axis targeted by an exchange command was not found in the source path (path where this exchange command was issued).

29 An axis specified in the exchange command is being processed by another command or has already been removed.
30 An axis targeted by an exchange command was not found in the destination path (path paired with a path where another exchange command was issued for the axis).
32 The exchange command has no target axis.
33 The exchange command has a conflict.
34 The exchange command has no axis specification or has an ID specification.
35 A cycle other than flexible path axis assignment is under way.
36 An attempt was made to perform flexible path axis assignment during the SV rotation control mode.
37 An attempt was made to perform flexible path axis assignment during the polygon turning mode.
38 An attempt was made to perform flexible path axis assignment during PMC axis control.
40 An attempt was made to perform flexible path axis assignment during mirror imaging.
41 An attempt was made to perform flexible path axis assignment during 3-dimensional coordinate conversion.
42 An attempt was made to perform flexible path axis assignment during coordinate system rotation.
43 An attempt was made to perform flexible path axis assignment during scaling.
44 An attempt was made to perform flexible path axis assignment during axis synchronization.
45 An attempt was made to perform flexible path axis assignment for an axis already removed.
46 An attempt was made to perform flexible path axis assignment for an axis under composite control.
47 An attempt was made to perform flexible path axis assignment for an axis under synchronous control.
48 An attempt was made to perform flexible path axis assignment for an axis under superimposed control.
55 An attempt was made to perform flexible path axis assignment simultaneously with an axis move command.
56 An attempt was made to perform flexible path axis assignment during tool compensation.
60 Axis number of axis removal or axis assignment exceed the limitation.
61 An attempt was made to perform Flexible path Axis Assignment in reset operation, or an attempt was made to perform Flexible path Axis Assignment in backward movement or re-forward movement of Manual Handle Retrace, or Setting of reference axis of the path cannot be changed, or An attempt was made to perform Flexible path Axis Assignment for moving axis, or Command for target axis exists (prepared by automatic operation) or Command for target axis exists (prepared by manual operation).
64 An attempt was made to perform Flexible path Axis Assignment for an axis under angular axis control.
65 An attempt was made to perform Flexible path Axis Assignment for an axis under flexible synchronous control.
66 An attempt was made to perform Flexible path Axis Assignment for an axis under EGB control.
70 The number of look ahead block is larger than original path which axis belongs.
71 The axis which belongs to loader path of Loader control function cannot be the target of Flexible path axis assignment.
72 Target path is in reset operation.
73 An attempt was made to perform Flexible path Axis Assignment in backward movement or re-forward movement of Manual Handle Retrace

74 An attempt was made to perform Flexible path Axis Assignment in backward movement or re-forward movement of Retrace.
75 Command for target axis exists (prepared by manual operation)
76 Command for target axis exists (prepared by manual operation)
77 Target axis is moving
78 Target axis is moving
79 Target axis is moving
80 Target axis is moving
81 Command for target axis exists (prepared by automatic operation)

4001	Belonging path of axis in flexible path axis assignment

A path (specified by parameter No. 981) to which an axis specified for flexible path axis assignment belongs is displayed.

0 : Source path
1 to 10 : Destination path (because of assignment or exchange)
-1 to -10 : Already removed

Parameter

0981	Absolute path number to which each axis belongs

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 1 to 10

Set the path to which each axis belongs.

> **NOTE**
> When 0 is set, each axis is assumed to belong to path 1.

11555	Flexible path axis assignment specified axis name

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 256

When the axis selected by bit0 (PAN) of parameter No.11564 is assigned, the specified axis name is used.

11556	Flexible path axis assignment specified axis name 2

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 256

When the axis selected by bit0 (PAN) of parameter No.11564 is assigned, the specified axis name is used. If the extended axis name is effective (bit0 (EEA) of parameter No.1000=1), this value becomes the second character of the axis name. Otherwise, this value becomes the subscript of the axis name.

11557	Flexible path axis assignment specified axis name 3

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 256

When the axis selected by bit0 (PAN) of parameter No.11564 is assigned, the specified axis name is used. If the extended axis name is effective (bit0 (EEA) of parameter No.1000=1), this value becomes the third character of the axis name. When the second axis name(No.11556) is not set, the third axis name becomes invalid.

11560	Identification number for an axis to be subjected to flexible path axis assignment

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 32767

Set an identification number for an axis to be subjected to flexible path axis assignment. The value specified corresponds to a program-specified address P(Q,R) value.

	#7	#6	#5	#4	#3	#2	#1	#0
11561				FAC	FAM	FAO	FAW	FAR

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 FAR The flexible path axis assignment function is:
0: Disabled.
1: Enabled.

> **NOTE**
> Setting the parameter FAR erases all history data (operation history, alarm history, and external operator message history) the next time the power is turned on.

#1 FAW If an axis acquisition command is issued for an axis yet to be freed in flexible path axis assignment:
0: The command waits for the axis to be freed.
1: Alarm PS0514 is issued.

#2 FAO If the power is turned off and on again with flexible path axis assignment in effect, the axis configuration is:
0: Returned to the initial state (specified with parameter No. 0981).
1: Kept in the most recent state.

#3 FAM The method of command specification used by programs in flexible path axis assignment is:
0: Identification number method.
1: Axis name method.

#4 FAC If the axis removement command is issued for an axis which already removed, or assigned to the another path in flexible path axis assignment:
0: Alarm PS0514 is issued..
1: The command is ignored.

	#7	#6	#5	#4	#3	#2	#1	#0
11562						FHE	FAN	

[Input type] Parameter input
[Data type] Bit axis

#1 FAN In flexible path axis assignment, axis names used after exchange are:
0: Those previously set for each axis.
1: Those set for the other axes in exchange pairs.

> **NOTE**
> The parameter FAN is invalid when an axis assignment command for flexible path axis assignment is issued. The name set for each axis is inherited.

#2 FHE In flexible path axis assignment, when parameter FHM(bit 3 of parameter No.11565) is 1, this axis :
0: Does not assign to the path which has large number of read ahead blocks more than initial path which the axis belongs.
1: Assigns to the path which has large number of read ahead blocks more than initial path which the axis belongs.

	#7	#6	#5	#4	#3	#2	#1	#0
11563				FAX				

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#4 FAX In flexible path axis assignment, if the target axis of the assignment command is already exists on the target path,
0: The alarm PS0514 occurs.
1: the command is ignored.

	#7	#6	#5	#4	#3	#2	#1	#0
11564								PAN

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

 # 0 PAN The name of the axis which assigned to each path is
0: Not changed.
1: Changed to the specific name selected in each path.
This setting is available on only one axis in a path.

	#7	#6	#5	#4	#3	#2	#1	#0
11565			FPS	FES	FHM			

[Input type] Parameter input
[Data type] Bit

#3 FHM On Flexible Path Axis Assignment, the axis which assigned to the path which has large number of read ahead blocks compared to the initial path which the axis belongs is:
0: selected automatically.
1: selected by parameter bit 2(FHE) of parameter No.11562.

#4 FES For the axis which the order in the path is changed by Flexible path axis assignment, Extended external machine zero point shift is:
0: not used.
1: used. (R address of Extended external machine zero point shift is not changed by Flexible path axis assignment.)

#5 FPS When the axis order is changed by Flexible path axis assignment, axis selection of position switch and high-speed position switch is:
0: depends on axis order after the execution of Flexible path axis assignment.
1: always considered as initial axis order.

Alarm and message

Number	Message	Description
PS0514	ILLEGAL COMMAND IN FLEXIBLE PATH AXIS ASSIGNMENT	1) An assignment command in flexible path axis assignment was issued for an axis not yet to be removed. 2) The P, Q, R, I, J, K, or L value specified by G52.1, G52.2, or G52.3 is invalid. 3) The value of the parameter No. 11560 is duplicated. 4) An attempt was made to execute a removal command (G52.1) for an axis already removed. 5) An attempt was made to exchange axes having different settings of bit 1 (FAN) of parameter No. 11562. 6) An attempt was made to perform flexible path axis assignment without canceling the offset. Refer to the diagnosis data No.4000 for more detailed cause of this alarm.

Caution

> ⚠ **CAUTION**
> Using this function changes the arrangement of axes, thus possibly resulting in an unintended axis (axis not intended by a parameter involving axis specification) being specified. When using this function, take the arrangement of axes into consideration.

> **NOTE**
> When the option is available, synchronous control and composite control are usable.

Limitation

1. This function cannot be used with Manual Guide *i*.
2. This function cannot be used with Program Restart.
3. This function cannot be used between an inch machine and a metric machine.
4. Flexible path axis assignment cannot be performed during the following modes.
   - Canned cycle
   - Coordinate system rotation
   - Mirror image
   - Scaling
   - Spindle control with servo motor
   - Polygon turning
   - High precision oscillation function
5. Do not specify any axis as a target in flexible path axis assignment if it is being used by any of the following:
   - Tandem control
   - Electronic gear box
   - Angular axis control
6. In retrace and manual handle retrace, it is impossible to reverse blocks in which flexible path axis assignment is specified.
7. When the setting of bit 1 (FAN) of parameter No. 11562 is valid, do not attempt axis exchange between an axis using an extended axis name and one using no extended axis name.

Notes concerning the FOCAS2 functions

Explained below are the notes concerning FOCAS2 functions. Observing the notes will make applications use flexible path axis assignment.

- Number of controlled axes in a path

The number of controlled axes in a path can be acquired using a FOCAS2 function (cnc_axisnum). The flexible path axis assignment function may change the number of controlled axes in a path. Therefore, acquire the number of controlled axes in a path as needed

- Information about controlled axes

Information about controlled axes can be acquired using a FOCAS2 function (cnc_rdaxisdata), which reads various data relating to the servo axis or spindle. The flexible path axis assignment function may change the configuration of controlled axes. In such a case, however, it is possible to acquire correct data. The FOCAS2 function can acquire information also about axes removed by flexible path axis assignment.

- History data

In the alarm history and the workpiece offset data change history, the number of controlled axes was changed as stated below according to whether the flexible path axis assignment function is enabled or disabled:

When the function is disabled: Intra-path relative axis number (1 to the allowable maximum number of controlled axes in a path)

When the function is enabled: Absolute axis number + 1000 (1001 to the allowable maximum number of controlled axes + 1000)

For details, refer to “FOCAS2 Function Reference”.

In addition, when acquiring axis names in the order of absolute axis numbers, use the function cnc_exaxisname2, which acquires extended axis names.

1.14.1 Outputting States of Individual Axes

Information about the path to which each axis belongs, their intra-path axis numbers, and whether they have been removed is output to a specified internal relay user area (R). Output information for each axis uses the 3 bytes in the internal relay user area (R) whose address is specified for them. Information output is carried out only for axes for which an internal relay user area (R) address has been specified. The output data is detailed below.

Path to which each axis belongs AXPT1<RXXXX.0> to AXPT4<RXXXX.3>

[Classification] Output signal

[Function] The path number of a path to which the axis of interest belongs is output. When using multi path PMC, only 1st PMC is used.

> **NOTE**
> XXXX represents a value set in parameter No. 11554.

Intra-path axis number for each path AXNM1<RXXXY.0> to AXNM5<RXXXY.4>

[Classification] Output signal

[Function] An axis number in the path to which the axis of interest belongs is output. When using multi path PMC, only 1st PMC is used.

> **NOTE**
> XXXY represents a value specified in parameter No. 11554 plus 1.

Whether each axis has been removed AXRMV<RXXXZ.0>

[Classification] Output signal

[Function] Information about whether each axis has been removed is output. When using multi path PMC, only 1st PMC is used.
Removed: 1
Assigned: 0

> **NOTE**
> XXXZ represents a value specified in parameter No. 11554 plus 2.

Signal address

XXXX represents a value specified in parameter No.11554.
XXXY represents a value specified in parameter No.11554 + 1.
XXXZ represents a value specified in parameter No.11554 + 2.

	#7	#6	#5	#4	#3	#2	#1	#0
RXXXX					AXPT4	AXPT3	AXPT2	AXPT1
RXXXY				AXNM5	AXNM4	AXNM3	AXNM2	AXNM1
RXXXZ								AXRMV

Parameter

11554	Internal relay user area (R) address for individual-axis information

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Valid data range] 0 to 59999

Set an internal relay user area (R) address for information to be output about an individual axis. The information is output only about the axis specified with this parameter. Three bytes starting at the setting are used for each axis.

NOTE
1 The setting must be a multiple of 4 (4, 8, ...).
2 This function is disabled if the parameter is 0.
3 When performing multipath control, be careful to keep the data addresses of each path from overlapping with those of the other paths.
4 The R address area varies depending on the PMC used and its memory. Be sure to select values within the usable range by checking the specifications of the PMC. (Example: R addresses in the range from R0 to R7999 if memory B of the first PMC is used.)

⚠WARNING
If an internal relay of the set address in this parameter is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the conflict of the used internal relay.

Example)
If parameter No. 11554 (C) = 1000, the output data of flexible path axis assignment is listed below.

Axis configuration	Path to which the C-axis belongs R1000	Intra-path axis number for the C-axis R1001	Whether the C-axis has been removed R1002
Path1: X Z C Y Path2: X Z Y	00000001	00000011	00000000
Removing the C-axis			
Path1: X Z Y C Path2: X Z Y	00000001	00000100	00000001

Axis configuration	Path to which the C-axis belongs R1000	Intra-path axis number for the C-axis R1001	Whether the C-axis has been removed R1002
Assigning the C-axis to path 2			
Path1: X Z Y Path2: X Z Y C	00000010	00000100	00000000

1.14.2 Direct Assignment Mode of Flexible Path Axis Assignment by signals

OVERVIEW

When the bit 0(CSG) of the parameter No.11563 is set to 1, direct assignment can be commanded by signal command with direct assignment mode signal DASN<G0536.5>.

Description

Direct assignment mode of Flexible path axis assignment

When the bit 2(NMF) of the Parameter No.11563 is set to 1, the direct assignment mode signal DASN<G0536.5> becomes available. If this signal is set to 1, the axis removal command, axis assignment command of flexible axis assignment can be commanded. Therefore, flexible path axis assignment command can be used even in the following situation (for example, command from machine operator panel).
In Direct assignment mode of Flexible path axis assignment, the axis assignment and axis removal is work as follows.

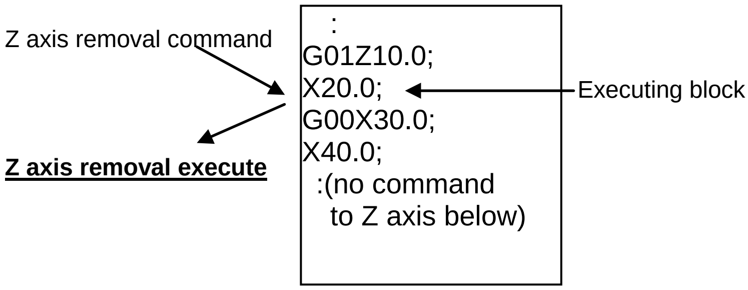
- Removal of control axis

When direct assignment mode signal DASN<G0536.5> and removal command signal RMVST<Gn536.2> are set to “1”, removal command is executed. When removal command is completed, Removal completion signal RMVED<Fn536.2> is set to “1”. When Removal command signal RMVST is set to “0”, Removal completion signal RMVED changes into “0”. The specification of the removal axis is selected by setting bit 0 of the address in user area of internal relay(R) (No.11553) to “1”. The removal command is only effective to the axis that belongs in the same path. It is possible to remove three axes at the same time.

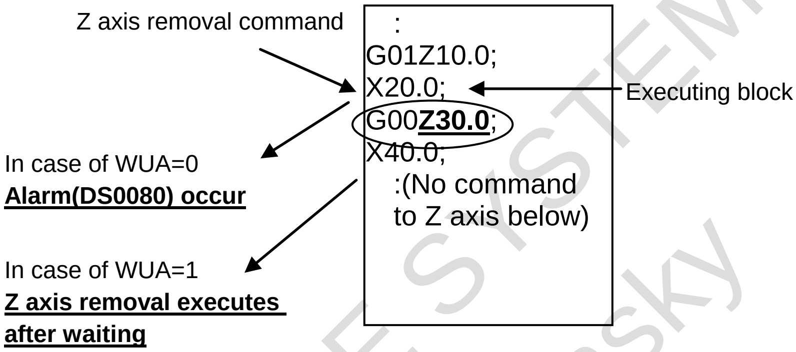
When axis removal is commanded, the CNC is worked as follows.

1. In case of the bit 3(WUA) of the Parameter No.11563 is 0 and any of the following item is applied, the alarm (DS0080) issues immediately and axis removal is not executed.
   1-1. The signal DASN<G0536.5> has changed while executing axis removal command.
   1-2. The axis removal is commanded while reset operation.
   1-3. The axis removal is commanded while emergency stop.
   1-4. The axis removal is commanded to the moving axis.
   1-5. The axis removal is commanded while AI Contour control mode, and any axis in the path is moving.
   1-6. The unit of reference axis (Parameter No. 1031) will change by the axis removal command.
2. In case of WUA(No11563#3) is 1, the CNC waits until item 1-1 to 1-5 becomes irrelevant.
   Keep the removal start signal and specified removal axis signal during waiting.
   The axis is removed when not all of above item 1-1 to 1-6 applies.

Example 1: In case of removal command is executed to Z axis while automatic operation with no Z axis command



Example 2: In case of removal command is executed while the NC is buffering automatic operation with Z axis command



- Assignment of control axis

When direct assignment mode signal DASN<G0536.5> and assignment command signal ASNST<Gn536.3> are set to “1”, assignment command is executed. When assignment command is completed, Assignment completion signal ASNED<Fn536.3> is set to “1”.
When Assignment command signal ASNST is set to “0”, Assignment completion signal ASNED changes into “0”.
The specification of the assignment axis is selected by setting bit 1 of the address in user area of internal relay(R) (No.11553) to “1”, which area is set for each axis. The axis order in the path when the assignment of control axis is commanded is specified by the address from bit 0 to bit 4 in user area of internal relay (R) ((No.11553)+1).
The path from which the control axis is assignment is specified by the address from bit 0 to bit 3 in user area of internal relay(R) ((No.11553)+2).
The assignment command is effective to the axis that belongs in the M code commanded path. Maximum three axes can be assigned at the same time.

When axis assignment is commanded on the path which is not in automatic operation, the axis is assigned immediately. In case of target axis is not removed, the axis assignment is not executed until the axis is removed.
In case of the automatic operation, if no buffered program command exists and every axis which belongs to the assignment path Cs contour controlling axis and servo live tool axis are not moving, the axis assignment is executed. If not, the alarm (DS0080) issues.
Therefore, in case of axis assignment command, start assignment sequence at the assigned path by suppress buffering M code to avoid the alarm above.

When axis assignment is commanded, if any of the following item is applied, the alarm (DS0080) issues immediately and axis assignment is not executed.

- The signal DASN<G0536.5> has changed while executing axis assignment command.

- The axis assignment is commanded while reset operation.
- The axis assignment is commanded while emergency stop.
- The unit of reference axis (Parameter No. 1031) will change by the axis removal command.
- The axis assignment is commanded while program is buffered.

NOTE

1. When the axis assignment command and removal command is executed to the same path, the preceded command is executed first. The next command is executed when the preceded command is finished and the assignment completion signal / removal completion signal becomes 0.
2. When executing macro blocks or executing short length movement blocks, the axis assignment / removal may take time.

-Specification of the order intra-path number of axis on axis assignment command

When no order number or illegal order number of the axis is specified by PMC signals, the CNC operation is as follows.

Ex) Y-axis and C-axis are assigned to the path of the axis configuration X-axis and Z-axis:
Setting value of Parameter No.11560: (X, Y, Z, C) = (1, 2, 3, 4),
No.11553: (X, Y, Z, C) = (100, 104, 108, 112)

1) In case that the axis order is not specified
Command: Assignment command ASNST <Gn536.3>=1 R104.1=1, R112.1=1, R105 = 0, R113 = 0
Y is assigned as the third axis, and C is assigned as the fourth and last axis. (X, Z, Y, C)
2) In case that the axis order is specified doubly.
Command: Assignment command ASNST <Gn536.3>=1 R104.1=1, R112.1=1, R105 = 2, R113 = 2
As "R104=2" and "R113=2" for second axis are specified doubly, the alarm (PS0514) is issued.
3) In case that the command contradicts axis configuration.
Command: Assignment command ASNST <Gn536.3>=1 R104.1=1, R112.1=1, R105 = 2, R113 = 5
As "R113=5" for fifth axis specification is contradicted, the alarm (PS0514) is issued.

Sample program

Example 1:

We show an example of axis assignment in case that M40 is a command for activating axis assignment ladder sequence M code by direct assignment mode signal DASN<G0536.5>.

If execute MDI operation at only one path (The rest of paths which concern to flexible path axis assignment are not executing automatic operation (Reset status)), the M code can be commanded in any path.

```
MDI program in path 1
O0000;
M40;
```

In case of automatic operation, the M code which activates the ladder sequence must be command from the assigned path. The sample program below shows the example that the axis removed from 2nd path and assigned to 1st path. By M40, the direct assignment mode signal is changed to 1, and then the ladder sequence that removes the axis from 2nd path is activated. Then the ladder sequence that assigns the axis to 1st path is activated. At last, the direct assignment mode signal is changed to 0.

(M40 is the M code which suppress buffering)

```
Path 1:
O1001
  :
M40
  :
```

```
Path 2:
O2001
  :
(Waiting M code is not necessary)
  :
```

Example 2:
This sample program shows that in case of the machine has following axis configuration and use the A axis from both 1st path and 2nd path.

1st path axis configuration: X1, Z1, C1, A
2nd path axis configuration: X2, Z2, C2
Parameter No.11553 = 100 for A axis

M104: Change the signal DASN<G0536.5> to 1, then activate the ladder sequence that remove the A axis from 2nd path. Then activate the ladder sequence that assign A axis to 1st path.
At last, change the signal DASN to0 and return FIN signal.

M204: Change the signal DASN<G0536.5> to 1, then activate the ladder sequence that remove the A axis from 1st path. Then activate the ladder sequence that assign A axis to 2nd path.
At last, change the signal DASN to 0 and return FIN signal.

(Both M104 and M204 are the M code which suppress buffering)

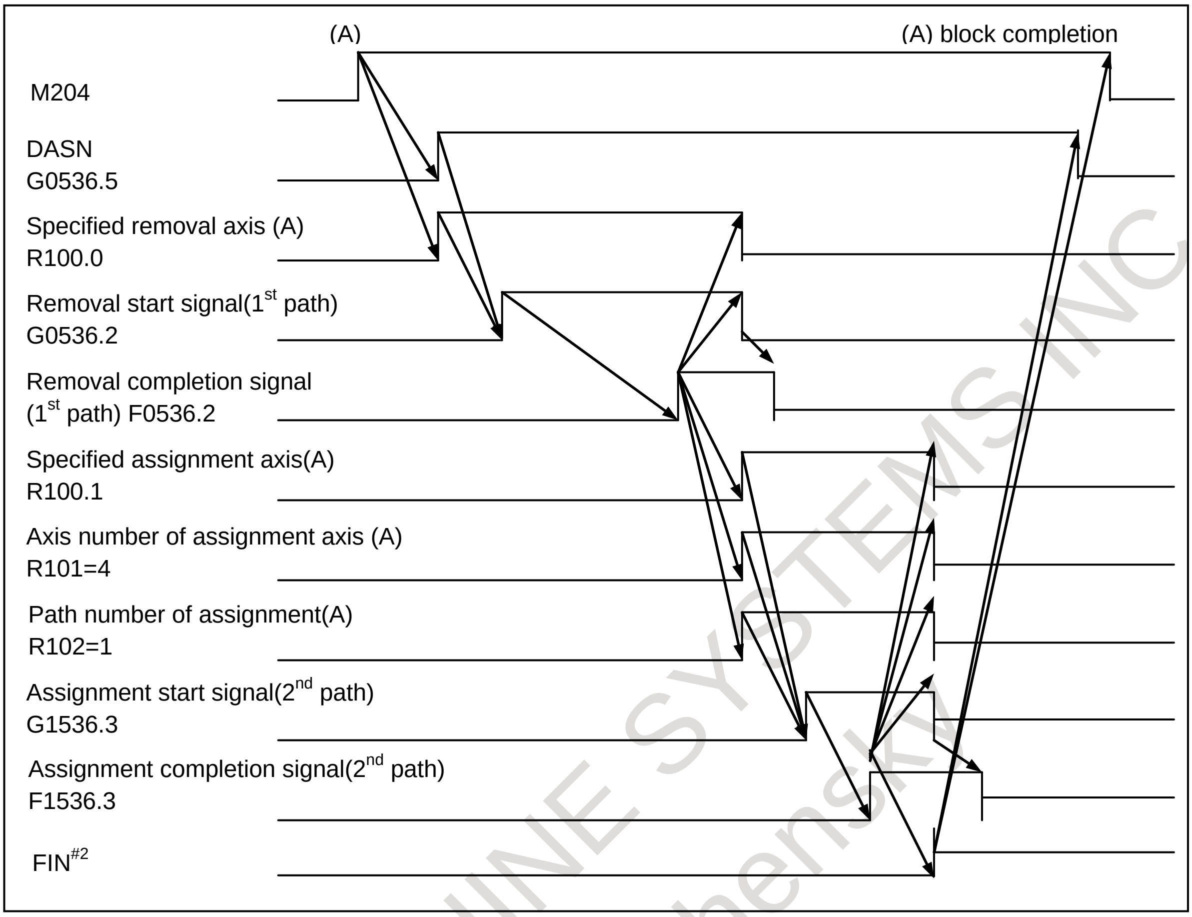
```
1st path
O1001;
  :
G01X1=100.0;                  (A')
G00X1=150.0;
G00Z1=230.0;
  :
M104;                         (B)
A10.0;
  :
```

```
2nd path
O2001;
  :
M204;                         (A)
A10.0;
  :
  :
G01X2=100.0;                  (B')
X2=110.0Z2=110.0;
X2=120.0;
G00X2=200.0Z2=200.0;
  :
```

(A)-(A’) Activate the ladder sequence from 2nd path that remove the A axis from 1st path and assign to 2nd path (M204)

(B)-(B’) Activate the ladder sequence from 1st path that remove the A axis from 2nd path and assign to 1st path (M104)

(Signal sequence on (A))



Signals

Removal start signal RMVST<Gn536.2>

[Classification] Input signal

[Function] Start removal command of flexible path axis assignment.

[Operation] When the signal is set to "1" with M code (Parameter No.11550), the removal command is executed.

Assignment start signal ASNST<Gn536.3>

[Classification] Input signal

[Function] Starts assignment command of flexible path axis assignment.

[Operation] When the signal is set to "1" with M code (Parameter No.11551), the assignment command is executed.

Direct assignment mode signal DASN<G0536.5>

[Classification] Input signal

[Function] Starts direct assignment mode of flexible path axis assignment.

[Operation] When the signal is set to "1", the signal type of flexible path axis assignment can be executed in case that the bit 2(NMF) of parameter No.11563 is 1.

Removal completion signal RMVED<Fn536.2>

[Classification] Output signal

[Function] Notifies the completion of removal command.

[Output condition] 0 : Removal command is not completed or removal command is not commanded.
1 : Removal command is completed.

When Removal command signal RMVST is set to “0”, this signal changes into “0”.

Assignment completion signal ASNED<Fn536.3>

[Classification] Output signal

[Function] Notifies the completion of assignment command.

[Output condition] 0 : Assignment command is not completed or assignment command is not commanded.
1 : Assignment command is completed.
When Assignment command signal ASNST is set to “0”, this signal changes into “0”.

- User area of internal relay (R) for each axis

The address in user area of internal relay (R) of each axis used for flexible path axis assignment is set to the parameter (No.11553). This parameter is used by three bytes an axis. Please set not to overlap with other axes.

The meaning of user area of internal relay (R) of each axis is as follows.

Specified removal axis RMVAX<RXXXX.0>

[Classification] Input signal

[Function] This signal specifies the removal axis. When using multi path PMC, only 1st PMC is used.

> **NOTE**
> XXXX represents a value set in parameterNo.11553.

Specified assignment axis ASNAX<RXXXX.1>

[Classification] Input signal

[Function] This signal specifies the assignment axis. When using multi path PMC, only 1st PMC is used.

> **NOTE**
> XXXX represents a value set in parameterNo.11553.

Axis number of assignment axis ASNM1<RXXXY.0> to ASNM5<RXXXY.4>

[Classification] Input signal

[Function] These signals specify the axis number of assignment axis in the path. When using multi path PMC, only 1st PMC is used.

> **NOTE**
> XXXY represents a value specified in parameter No. 11553 plus 1.

**Path number of assignment or path number of exchange target axis
CMPT1<RXXXZ.0> to CMPT4<RXXXZ.3>**

[Classification] Input signal

[Function] These signals specify the path number of assignment command.
These signals specify the path number of exchange target axis. When using multi path PMC, only 1st PMC is used.

> **NOTE**
> XXXZ represents a value specified in parameter No. 11553 plus 2.

Signal address

Gn536			DASN		ASNST	RMVST		
Fn536					ASNED	RMVED		

- User area of internal relay (R)

XXXX represents a value set in parameter No.11553.
XXXY represents a value specified in parameter No. 11553 plus 1.
XXXZ represents a value specified in parameter No. 11553 plus 2.

RXXXX							ASNAX	RMVAX
RXXXY				ASNM5	ASNM4	ASNM3	ASNM2	ASNM1
RXXXZ					CMPT4	CMPT3	CMPT2	CMPT1

Parameter

11553	The address of command in user area of internal relay(R)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Valid data range] 0 to 59999

When the signal type of flexible path axis assignment is used, the address of command in user area of internal relay (R) is set in each axis. 3 bytes are used from a set value with each axis.

NOTE
1 Set the value that becomes the multiple of 4. (0, 4, 8, …)
2 The range of the R address differs depending on the PMC used and the memory size. Check the specifications of the PMC, and set a value within the valid range.
(Example: R addresses in the range from R0 to R7999 if memory B of the first PMC is used.)
3 Signal type of flexible path axis assignment is not available for the axis in case of this parameter is set to 0.

⚠WARNING
If an internal relay of the set address in this parameter is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used internal relay.

	#7	#6	#5	#4	#3	#2	#1	#0
11563					WUA	NMF		CSG

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 CSG Flexible path axis assignment is commanded:
0: By G code command.
1: By PMC signal setting.

> **⚠WARNING**
> Please set an appropriate value to parameter No.11553 beforehand when you set 1 to this parameter.
> If an internal relay of the set address in parameter No.11553 is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used internal relay.

 # 2 NMF Direct assignment mode signal DASN<G0536.5> is:
0: Not used.
1: Used.

 # 3 WUA In direct assign mode of the flexible path axis assignment, if removal command could not execute,
0: The alarm (DS0080) occurs.
1: Wait execution of the axis removal.

Alarm and message

Number	Message	Description
DS0080	ILLEGAL COMMAND IN FLEXIBLE PATH AXIS ASSIGNMENT	1) The program is buffered. 2) Signal DASN<G0536.5> is changed. 3) Reset operation is executed. 4) The machine is in emergency stop state. 5) Axis exchange command is executed in the direct assignment mode 6) The unit of the reference axis cannot be changed. 7) Target axis is moving Refer to the diagnosis data No.4000 for more detailed cause of this alarm.

Limitation

- Direct assignment of Flexible path axis assignment cannot execute during rigid tapping mode.
- Axis assignment and axis removal cannot execute during backward movement, re-forward movement of Manual handle retrace function.
- Flexible path axis assignment cannot be executed during high precision oscillation.
  If you command as above, the alarm (DS0080) issues.

1.14.3 Axis Assignment Reset

OVERVIEW

When using Flexible path axis assignment, if specified parameter is set, axis assignment can be initialized automatically by reset operation.
When automatic operation is halted by reset operation, it becomes easy to initialize the axis assignment.

Description

When bit 6(RRS) of the parameter No. 11563 is set to 1, and every path which execute the Flexible path axis assignment becomes the reset state simultaneously by MDI reset key operation , external reset signal, or reset and rewind signal, axis assignment is initialized automatically at the completion of reset state.
The axis assignment is initialized or not can be verified by checking Initial axis assignment signal INIST <F0536.7>.

To initialize axis assignment, every axis which belongs to the target path must be the state which the Flexible path axis assignment can execute. If any axis is not, the initialize of axis assignment is not executed.
The state which the flexible axis assignment can execute is based on the specifications of Direct assignment of Flexible path axis assignment.

Following is the examples that axis assignment reset cannot be execute.

- Any axis which enables the Flexible path axis assignment is executing the PMC axis control on target path.
- Tool offset is enabled on target path, and the offset is not cancelled by reset operation.

The axis which is not assigned to another path might be target of axis removal/assignment by Axis assignment reset for restoration to original axis configuration. However, the axis that the Flexible path axis assignment is not enable (the parameter No.11560 = 0) is not becomes target of axis removal / assignment.
Example)
The machine has the original axis configuration that X1,Z1,C1 axes are on 1st path, and X2,Z2,C2 axes are on 2nd path. Flexible path axis assignment is enabled on Z1 and Z2 axis.

1) In case of axis assignment is changed by Flexible path axis assignment that X1, Z1, Z2, C1 on 1st path and X2, C2 on 2nd path.

   When both 1st and 2nd path becomes reset state simultaneously, if Z2 axis is able to execute Flexible path axis assignment, the 1st and 2nd path become initial axis assignment after reset.

2) In case of axis order is changed to X1, C1, Z1 in 1st path and no axis changed in 2nd path by Flexible path axis assignment.
   When reset is executed on 1st path and Z1 axis is able to execute Flexible path axis assignment, the 1st path becomes initial axis order after reset operation.

3) In case of axis order has changed to X1, Z2, C1 on 1st path, and X2, Z1, C2 on 2nd path by Flexible path axis assignment.
   When both 1st and 2nd path becomes reset state simultaneously, and both Z1, Z2 axis is able to execute Flexible path axis assignment, the 1st and 2nd path becomes initial axis assignment after reset.

Signals

Initial axis assignment signal INIST<Fn536.7>

[Classification] Output signal
[Function] Notifies the path has initial axis assignment or not.
[Output condition] 0: The axis assignment of the path is not same as initial state.

1: The axis assignment of the path is same as initial state.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11563		RRS						

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 6 RRS When reset is executed, the axis assignment changed by Flexible path axis assignment
0: Is not changed.
1: Is returned to its initial assignment.

Note

High precision oscillation function

When reset is executed during precision oscillation and this function is available, another path which is not in initial assignment keeps reset. The reset is finished when the precision oscillation axis is returned to R position and chopping mode is canceled.

External reset, Reset and Rewind

When this function is enabled, and reset and rewind or external reset is executed to each path individually, the axis assignment reset is executed when all target paths become the reset state.

1.14.4 Improvement of PMC Window Function with Flexible Path Axis Assignment

Outline

In flexible path axis assignment, it has been enabled to read and write the axis information by PMC window function according to the path number and axis number of the initial state specified by parameter (No.981).

Explanation

Behavior of PMC window function with flexible path axis assignment

In the conventional specification, PMC window function with flexible path axis assignment behaves as follows:

- When one axis is specified for read/write, the axis number and the path number changed by flexible path axis assignment should be specified.
- When all axes in a path are specified for read/write, target axes are determined by the path numbers of each axis changed by flexible path assignment. Furthermore, the order of each axis data and the axis number read by function code 433 (CNC alarm detail read) depend on axis number after flexible path assignment.
- The number of axes in a path acquired by function code 0 (reading CNC system information) depends on the path numbers of each axis after flexible path assignment.

If the bit 7(PPW) of parameter No.11563 is set to 1, behavior of PMC window function with flexible path axis assignment is not changed by the execution of flexible path assignment as follows:

- When one axis is specified for read/write, the axis number and the path number of the initial state can be specified.

- When all axes in a path are specified for read/write, target axes are determined by the initial path numbers of each axis. Furthermore, the order of each axis data and the axis number read by function code 433 (CNC alarm detail read) depend on the initial axis number.
- The number of axes in a path acquired by function code 0 (reading CNC system information) depend on the initial path numbers of each axis.

Example)

Flexible path axis assignment is executed as following table.

	Before flexible path axis assignment	
Axis number	**Path 1**	**Path 2**
1	X1	X2
2	Y1	Y2
3	Z1	Z2
4	A1	
5	**'C'**	

→

	After flexible path axis assignment	
Axis number	**Path 1**	**Path 2**
1	X1	X2
2	Y1	Y2
3	Z1	Z2
4	A1	**'C'**
5		

In this example, to read the data of C-axis, specify the axis number and the path number as following table.

		The axis number and path number specified to read the data of C-axis
Parameter PPW (No.11563#7)=0	Before flexible path axis assignment	5th axis in the path 1
	After flexible path axis assignment	4th axis in the path 2
Parameter PPW (No.11563#7)=1	Before flexible path axis assignment	5th axis in the path 1
	After flexible path axis assignment	

When all axes in the path is specified for read/write, target axes are shown in the following table.

		Target axes of read/write with all axes are specified	
		Path 1	**Path 2**
Parameter PPW (No.11563#7)=0	Before flexible path axis assignment	X1,Y1,Z1,A1,C	X2,Y2,Z2,A2
	After flexible path axis assignment	X1,Y1,Z1,A1	X2,Y2,Z2,A2,C
Parameter PPW (No.11563#7)=1	Before flexible path axis assignment	X1,Y1,Z1,A1,C	X2,Y2,Z2,A2
	After flexible path axis assignment		

The numbers of axes in each path are read by function code 0 (reading CNC system information) as shown in the following table.

		The numbers of axes in each path read by function code 0 (reading CNC system information)	
		Number of axes in path 1	**Number of axes in path 2**
Parameter PPW (No.11563#7)=0	Before flexible path axis assignment	5	3
	After flexible path axis assignment	4	4
Parameter PPW (No.11563#7)=1	Before flexible path axis assignment	5	3
	After flexible path axis assignment		

Supported PMC WINDOW functions

Supported PMC WINDOW functions are shown in following table.

Description	Function code	Response	R/W
Reading CNC system information	0	High-speed	R
Reading a work piece origin offset value	15	High-speed	R
Writing a work piece origin offset value	16	Low-speed	W
Reading a parameter	17,154	High-speed	R
Writing a parameter	18	Low-speed	W
Reading a real type parameter	321	High-speed	R
Writing a real type parameter	323	Low-speed	W
Reading setting data	19,155	High-speed	R
Writing setting data	20	Low-speed	W
Reading diagnosis data	33	Low-speed	R
Reading detailed information of CNC	433	Low-speed	R
Reading the absolute position (absolute coordinates) of controlled axes	27	High-speed	R
Reading the machine position (machine coordinates) of controlled axes	28	High-speed	R
Reading a skip position (stop coordinates of skip operation (G31)) of controlled axes	29	High-speed	R
Reading the servo delay for controlled axes	30	High-speed	R
Reading the acceleration/deceleration delay on controlled axes	31	High-speed	R
Reading the feed motor load current value (A/D conversion data)	34	High-speed	R
Reading the relative position on a controlled axis	74	High-speed	R
Reading the remaining travel on a controlled axis	75	High-speed	R
Reading the actual velocity of each controlled axes	91	High-speed	R
Entering torque limit data for the digital servo motor	152	Low-speed	W
Reading the actual speed on a controlled axis	207	High-speed	R
Reading the estimate disturbance torque data	211	High-speed	R
Presetting the relative coordinate	249	Low-speed	W
Reading the actual machine position of controlled axes	428	High-speed	R
Reading the command value of axes	446	High-speed	R

Limitation

Least input increment

When the improvement is valid (No.11563#7=1) and the path number of the axis is changed by flexible path axis assignment, to treat the data of this axis by PMC window function, least input increment of the current path must be same as least input increment of the initial path.

Example)

Flexible path axis assignment is executed as following table.

	Before flexible path axis assignment	
Axis number	**Path 1**	**Path 2**
1	X1	X2
2	Y1	Y2
3	Z1	Z2
4	A1	
5	**'C'**	

→

	After flexible path axis assignment	
Axis number	**Path 1**	**Path 2**
1	X1	X2
2	Y1	Y2
3	Z1	Z2
4	A1	**'C'**
5		

In this example, to execute read/write of the data of C-axis by PMC window function, least input increment of path 1 must be same as least input increment of path 2.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11563	PPW							

[Input type] Parameter input
[Data type] Bit

#7 PPW PMC window function with execution of flexible path axis assignment behaves according to:
0: The path number and the axis number changed by flexible path axis assignment.
1: The path number and the axis number of the initial state (setting of the parameter No.981).

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

1.15 HIGH PRECISION OSCILLATION FUNCTION

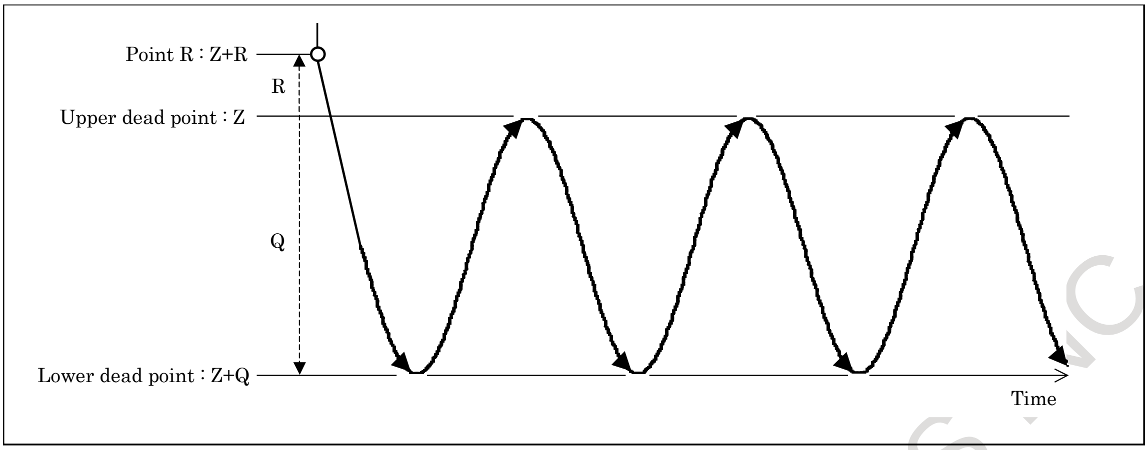
Overview

In this function, the feedrate of oscillation axis (equivalent to a chopping axis that is moved vertically and repeatedly for grinding) changes along sine curve. This function is effective to improve, the accuracy of movement between upper dead point and lower dead point.
Moreover, look-ahead feed forward function can be applied to oscillation motion. Then the servo delay can be almost eliminated and higher accuracy can be achieved even if oscillation feedrate or upper or lower dead point is changed.

This function becomes effective by setting the bit 0 (SSO) of parameter No.25650 is set to 1.

Format

G81.1 Z_ Q_ R_ F_ ;
Z : Upper dead point (In case that an axis is other than the Z-axis, specify the axis address. Specify the distance as an absolute value.)
Q : Distance between the upper dead point and lower dead point
(Specify the distance as an incremental value from the upper dead point.)
R : Distance from the upper dead point to point R
(Specify the distance as an incremental value from the upper dead point.)
F : Oscillation base feedrate
G80 ; Cancels oscillation



If the addresses of Z, Q, R, or F are omitted, the oscillation motion is performed by the value of parameters. On the other hand, the value of parameters is replaced by the value commanded for each address.

Z (Axis address)	Parameter (No.8370)	: Oscillation axis number = Axis number.
	Parameter (No.8372)	: Upper dead point = Value of address Z.
Q	Parameter (No.8373)	: Lower dead point = Value of address Q + value of address Z (If address Z is omitted, the value is parameter (No.8372)).
R	Parameter (No.8371)	: Point R = Value of address R + value of address Z (If address Z is omitted, the value is parameter (No.8372)).
F	Parameter (No.8374)	: Oscillation base feedrate = Value of address F.

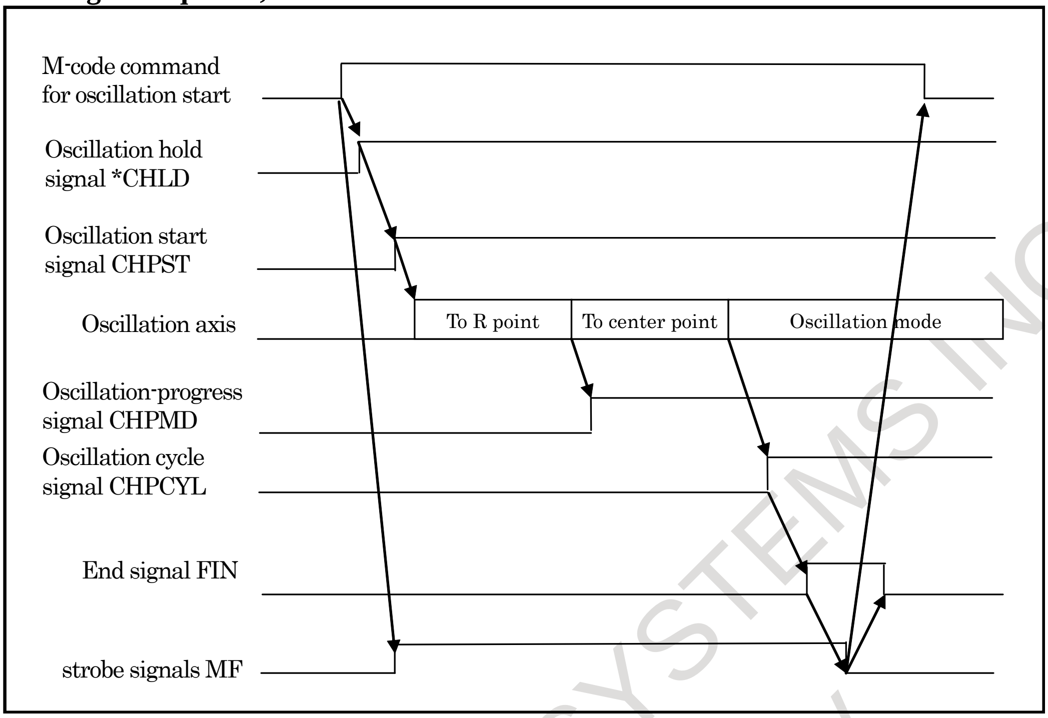
NOTE
1 G81.1/G80 is the G code for suppressing buffering.
2 Specify G81.1/G80 in a single block.

- Oscillation activated by signal input

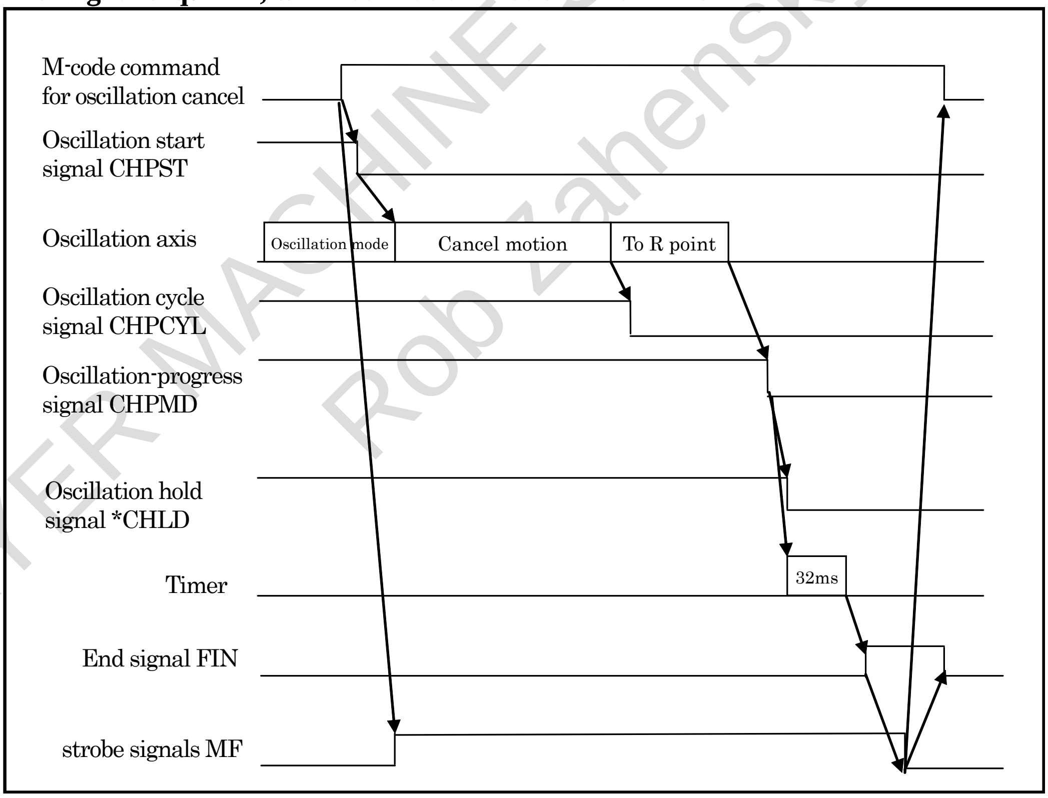
Before oscillation motion is started, oscillation axis, reference position, upper dead point, lower dead point, and oscillation base feedrate must be set on the parameter screen or the oscillation screen. Oscillation motion is started once oscillation start signal CHPST <Gn051.6> has been set to "1". This signal is ignored, however, while oscillation axis is moving during automatic operation.
When oscillation hold signal *CHLD <Gn051.7> is set to "0" during oscillation motion, the tool immediately moves to point R and stops. Setting the oscillation hold signal *CHLD to "1" restarts oscillation motion. Oscillation motion can also be stopped by setting oscillation start signal CHPST to "0", but only when oscillation motion was started by using that signal.

Method of starting oscillation	Method of cancelling oscillation	State
Signal CHPST ="1"	Signal *CHLD = "0"	It stops. (Suspending state)
	Signal CHPST = "0"	It stops. (Cancel state)
	G80 command	It stops. (Cancel state)
G81.1 command	Signal *CHLD = "0"	It stops. (Suspending state)
	Signal CHPST = "0"	It doesn't stop.
	G80 command	It stops. (Cancel state)

- Example of signal sequence, when oscillation motion is started



- Example of signal sequence, when oscillation motion is canceled



NOTE

1. Switching to manual mode or suspending automatic operation, by means of feed hold, does not stop oscillation motion.
2. If a move command is specified for the oscillation axis while oscillation is being performed, an alarm (PS5050) is issued.
3. If a G81.1 command is specified during oscillation started by the signal, oscillation motion is not stopped. If point R, upper dead point, lower dead point, or oscillation feedrate has been modified by using the G81.1 command, oscillation is continued but using the modified data.
4. The use of oscillation start signal CHPST <Gn051.6> to start oscillation motion is not enabled immediately after power-on; it is not enabled until the completion of manual reference position return.
5. The M codes which are used to set the oscillation start signal CHPST <Gn051.6> to "1" or "0" should be specified as M code preventing buffering (parameter Nos. 3411 to 3432).

⚠**CAUTION**

After the cancel of oscillation motion during NC automatic operation, to execute a move command or a coordinate system setting command (in the absolute or incremental mode) for the oscillation axis in an NC block during the same operation, use G-code (G80) for the cancel of oscillation.

If it is necessary to cancel by signal input, execute the following (1) or (2).

(1) M code in which each axis workpiece coordinate system preset signal WPRST<Gn358> is turned ON
(Refer to 1.5.2.7"Each axis workpiece coordinate system preset signals" for details.)

(2) Workpiece coordinate system preset (G92.1)
(Enable workpiece coordinate system preset (G92.1) (bit 1 (NWC) of parameter No.8136 is 0).

For example, in the following program (O0001), to execute the move command (N60) after the cancel (N40) of oscillation motion (Z-axis), “M code in which each axis workpiece coordinate system preset signal WPRST is turned ON” or “Workpiece coordinate system preset (G92.1)” must be specified in N50.

```
O0001
N10 G94 G90 G00 X100.0
N20 M55                  ← Oscillation start signal is turned ON
N30 G01 X30.0 F30.0
N40 M56                  ← Oscillation start signal is turned OFF
                         ← “M code in which each axis workpiece
                            coordinate system preset signal WPRST for
N50 ____                    Z-axis is turned ON” or “Workpiece
                            coordinate system preset (G92.1Z0.0)” is
                            commanded.
N60 G00 Z-50.0           ← Move command of Z-axis
N70 M02
```

- Oscillation feedrate (feedrate of movement to point R)

From the start of oscillation motion to reaching point R, the tool moves at the rapid traverse rate (specified by parameter No.1420).

The override function can be used for either the normal rapid traverse rate or oscillation feedrate, one of which can be selected by setting bit 0 (ROV) of parameter No.8360. When the oscillation feedrate is overridden, settings more than 100% are clamped to 100%.

- Oscillation feedrate (feedrate of movement from point R)

From point R, where reached after the start of oscillation motion, to the center point between upper and lower dead points, the tool moves at the oscillation base feedrate (F). Afterwards, between the center point and the point where the oscillation motion is canceled, the tool moves with sine curve feedrate by Exp. 1.

$$f(t)_{[mm/\min]} = k \times \mathrm{F}_{[mm/\min]} \times \sin\left(\frac{2}{|\mathrm{Q}|_{[1/mm]}} \times \frac{k \times \mathrm{F}}{60}_{[mm/\sec]} \times \frac{180}{\pi}_{[\deg/rad]} \times t_{[\sec]} \right) \qquad \ldots \text{Exp. 1}$$

F(t) : Sine curve feedrate [mm/min]
F : Oscillation base feedrate [mm/min]
Q : Distance between the upper dead point and lower dead point [mm]
k : Oscillation override (0.0 (0%) to 1.5 (150%))
Set by oscillation federate override signals *CHP1 - *CHP8<Gn051.0-3>
t : Time [sec]

The value (*k*F) of the product of oscillation base feedrate and oscillation override is clamped to the maximum oscillation feedrate (parameter No.8375) if the value (*k*F) is greater than the maximum oscillation feedrate. However, even if the value (*k*F) is not greater than maximum oscillation feedrate, it is clamped by the feedrate never exceeds the maximum allowable acceleration rate for acceleration/deceleration (parameter No.25652) during oscillation motion.

- Setting oscillation data

Set the following oscillation data:

Oscillation axis…………………Parameter (No.8370)
Reference point (point R)………Parameter (No.8371)
Upper dead point……………….Parameter (No.8372)
Lower dead point……………….Parameter (No.8373)
Oscillation base feedrate………..Parameter (No.8374)
Maximum oscillation feedrate… Parameter (No.8375)

All data items other than the oscillation axis and maximum oscillation feedrate can be set on the oscillation screen. Moreover, the value of parameters is replaced by the value of G81.1 command.

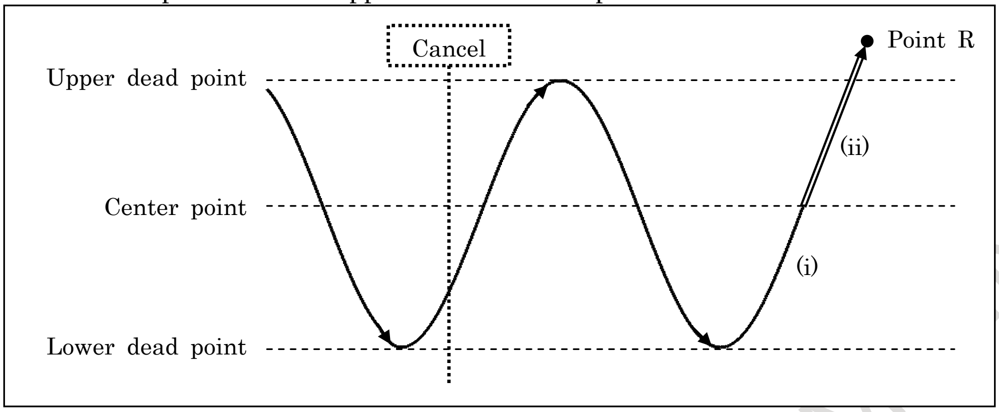
- Stopping oscillation

The following table lists the operations and commands that can be used to stop oscillation motion, the positions at which oscillation stops, and the operation performed after oscillation stops:

Operation/ command	Stop position	Operation after stops
G80 command	It follows parameter OST (No.25651#0). OST=0 : Point R OST=1 : Decelerated and stopped	Canceled
Reset		
PS, OT alarm		
CHPST: "0"	It follows parameters OST, SGS (No.25651#0,#2). OST=0 or SGS=0 : Point R OST=1 and SGS=1 : Decelerated and stopped	Canceled
*CHLD: "0"	It follows parameters OST, SGS, HST (No.25651#0,#2,#3). OST=0, SGS=0, or HST=0 : Point R OST=1, SGS=1, and HST=1 : Decelerated and stopped	Restart after *CHLD goes "1"
Emergency stop	The tool stops immediately.	Canceled
Servo alarm		

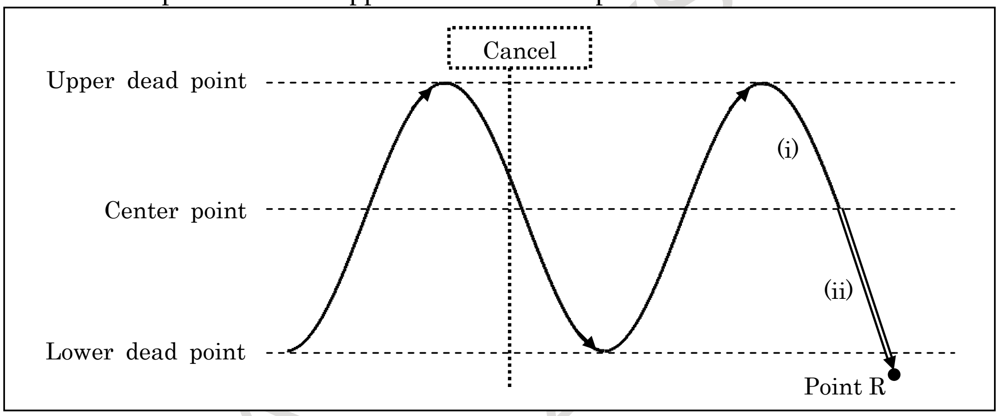
Example of stopping at point R. (Except in the case of suspending by signal *CHLD = 0 or issuing an OT alarm.)

(1) Point R > Center point between upper and lower dead points



After cancel command, oscillation motion is continued until oscillation axis passes center point between upper and lower dead points via next lower dead point (i).
Afterwards, the axis moves to point R by linear interpolation (ii).

(2) Point R < Center point between upper and lower dead points



After cancel command, oscillation motion is continued until oscillation axis passes center point between upper and lower dead points via next upper dead point (i).
Afterwards, the axis moves to point R by linear interpolation (ii).

In the case of suspending by signal *CHLD="0" or cancel by OT alarm, oscillation axis moves to point R at once.

NOTE

The timing of cancellation motion might change according to the position of point R or the setting of acceleration rate of cancellation motion (parameter No.25653).

- Movement from point R to the first dead point

The first dead point is decided by the position of point R.

Point R	The first dead point
Point R is on upper dead point side from center point or is on the center point.	Lower dead point
Point R is on lower dead point side from center point.	Upper dead point

- Omission of movements to point R and to the center point

By setting 1 to bit 0 (OST) and 1 (FFS) of parameters No.25651, if oscillation motion is started by G81.1 command, movements to point R and to the center point can be omitted, and oscillation motion can be immediately started. In this case, the first dead point is as follows.

State before G81.1 command	The first dead point
In case of the first command of oscillation motion after the power is turned on or in the state of reset.	Lower dead point
In the state that command of cancel was specified on the way to lower dead point at last oscillation motion.	
In the state that command of cancel was specified on the way to upper dead point at last oscillation motion.	Upper dead point

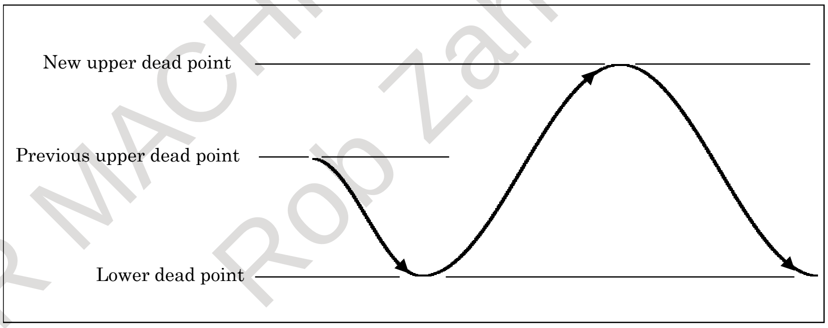
- Acceleration/deceleration

Linear acceleration/deceleration by parameter (No.25653) is applied to the movement from point R to center point or the motion till stop.
Moreover, when override is changed during oscillation motion, the acceleration/deceleration set by parameter (No.25653) is effective.
Acceleration/deceleration of sine curve feedrate control is enabled on oscillation motion.

- Oscillation after the upper dead point or lower dead point has been changed

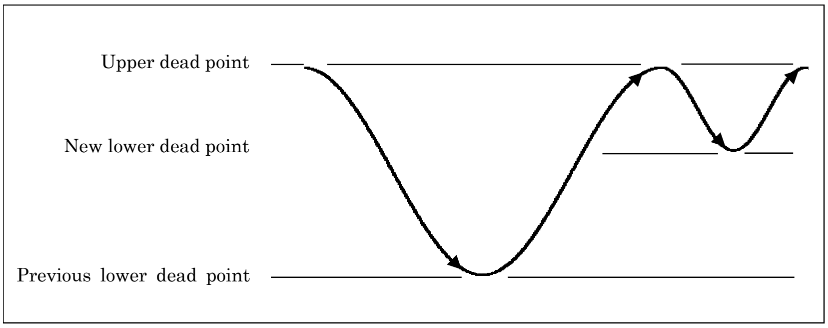
When the upper dead point or lower dead point is changed while oscillation motion is being performed, the tool moves to the dead points specified by the old data. Then, oscillation motion is changed to the new one using the new data.
The following describes the operations performed after the data has been changed.

(1) When the upper dead point is changed during movement from the upper dead point to the lower dead point



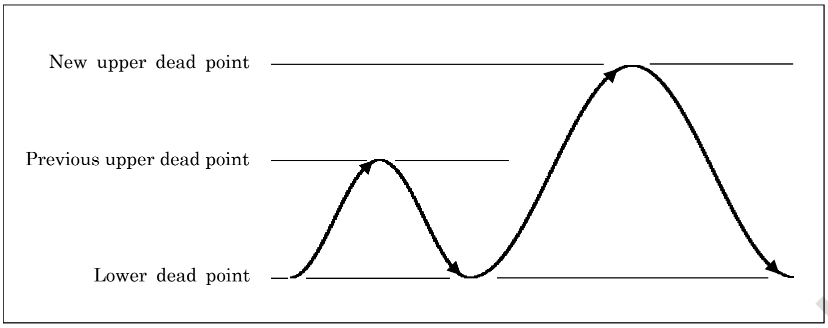
The tool first moves to the lower dead point, then to the new upper dead point.

(2) When the lower dead point is changed during movement from the upper dead point to the lower dead point



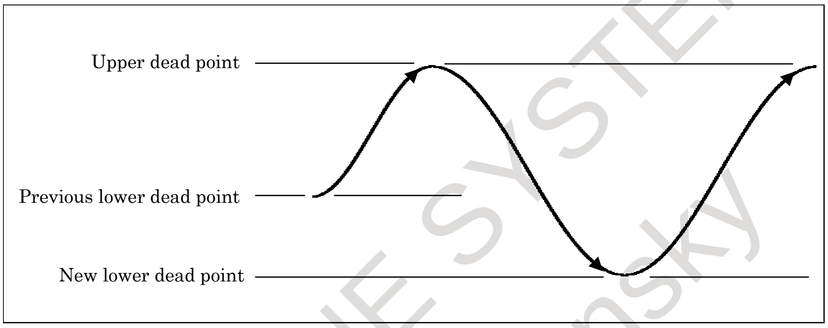
The tool first moves to the previous lower dead point, then to the upper dead point, and finally to the new lower dead point.

(3) When the upper dead point is changed during movement from the lower dead point to the upper dead point



The tool first moves to the previous upper dead point, then to the lower dead point, and finally to the new upper dead point.

(4) When the lower dead point is changed during movement from the lower dead point to the upper dead point



The tool first moves to the upper dead point, then to the new lower dead point.

- Look-ahead feed forward function

Look-ahead feed forward function can be effective for oscillation motion.
Refer to the following manual for details of the setting method and so on.

- FANUC AC SERVO MOTOR *αi* series PARAMETER MANUAL (B-65270EN)
  "Look-ahead feed forward function"

- Mode switching during oscillation motion

If the mode is changed during oscillation motion, oscillation motion does not stop. In manual mode, the oscillation axis cannot be moved manually. It cannot also be moved manually by means of the handle interrupt.

- Single block signal

Even when single block signal SBK<Gn046.1> is input during oscillation motion, oscillation continues.

- Interlock signal

Interlock signal for all axes *IT<Gn008.0> or Interlock signals for each axis *ITx<Gn130> is valid to the oscillation axis in oscillation mode. However, Interlock signal for each axis direction +MITx/-MITx <Gn132/Gn134, X004.2 to X004.5> is invalid to the oscillation axis in oscillation mode.
If interlock is effective to the oscillation axis during oscillation motion, the oscillation axis decelerates and stops immediately.

- Flexible synchronization control

Flexible synchronization control whose master axis is the oscillation axis can be used.
If Look-ahead feed forward function is used, it is necessary to set the parameter also to the slave axis as well as the master axis.
However, Inter-path flexible synchronization control with oscillation axis cannot be used.

- Axis synchronous control

Axis synchronous control whose master axis is the oscillation axis can be used.
If Look-ahead feed forward function is used, it is necessary to set the parameters also to the slave axis as well as the master axis.

- Machine lock during oscillation motion

"If all-axis machine lock signal MLK<Gn044.1> or each-axis machine lock signals MLK1 to MLK8<Gn108> are turned ON or OFF during oscillation movement, an alarm PS5050 "ILL-COMMAND IN G81.1 MODE" is issued.
After an alarm PS5050 is issued, oscillation axes move to the point R, then the axes become the machine lock state. "

- Diameter and Radius Setting

The command and the parameter value (coordinates, feedrate) follow the diameter and radius setting of the oscillation axis. Actual data value (feedrate F) is displayed by the radius.

Limitation

- Workpiece coordinate system

While oscillation is being performed, do not change the workpiece coordinate system for the oscillation axis.

- PMC axis

When the oscillation axis is operating as the PMC axis, oscillation is not started.

- Mirror image

Never attempt to apply the mirror image function about the oscillation axis.

- Move command during oscillation motion

If a move command is specified for the oscillation axis while oscillation motion is being performed, an alarm (PS5050) "ILL-COMMAND IN G81.1 MODE" is issued.

- Canned Cycle

While oscillation is being performed, do not specify canned cycle.

- Program restart / Quick program restart

When a program contains G codes for starting oscillation (G81.1) and stopping oscillation (G80), an attempt to restart that program results in an alarm (PS5050) being output.
When a program that does not include the oscillation axis is restarted during oscillation motion, the coordinates and amount of travel set for the oscillation axis are not affected after the restart of the program.

- Rotation axis

Oscillation operation cannot be used with rotation axis (type-A).

- Inch/Metric conversion commands

While oscillation is being performed, do not specifyinch/metric conversion commands.

- General purpose retract

While oscillation is being performed, do not perform retraction. The oscillation motion does not stop by retraction.

- Stored stroke check

Stored stroke check 1-I (parameter Nos. 1320 and 1321) is effective during oscillation motion.

- Arbitrary angular axis control

Do not set angular axis and perpendicular axis of arbitrary angular axis control to oscillation axis.

- Three-dimensional coordinate conversion / Tilted working plane indexing command

While oscillation is being performed, do not specify three-dimensional coordinate conversion / tilted working plane indexing command.

- Cs contour control axis

Do not set Cs contour control axis to oscillation axis. If oscillation is specified for Cs contour control axis, an alarm (PS5050) "ILL-COMMAND IN G81.1 MODE" is issued.

- Synchronous/Composite control

Do not set synchronous/composite control axis to oscillation axis.

- Polygon turning

Do not set control axis for polygon turning to oscillation axis.

- Manual handle interrupt

Manual handle interrupt is invalid to the axis in the oscillation mode.
(The slave axis of Flexible synchronization control which has oscillation axis as master or Axis synchronous control is contained)

- Constant surface speed control

Do not set reference axis for constant surface speed control to oscillation axis.

- Polar coordinate interpolation

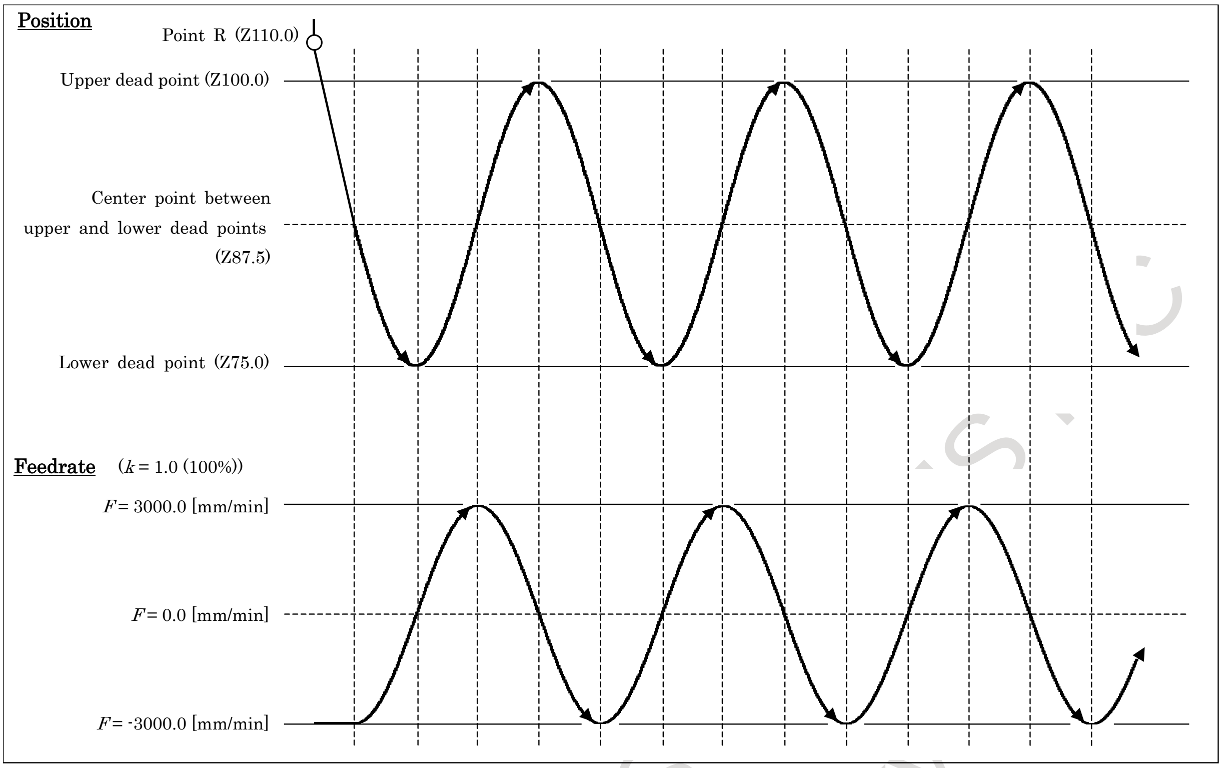
Do not set linear/rotation axis for polar coordinate interpolation to oscillation axis.

Example

To start oscillation, specify the following command:

G90 G81.1 Z100. Q-25. R10. F3000. ;

- The tool moves to point R (Z+R = Z110.0) with rapid traverse feedrate.
- Next, move to center point (Z87.5) between upper dead point (Z100.0) and lower dead point (Z+Q = Z75.0) at oscillation base feedrate (F3000.0 [mm/min]).
- Then, perform repeated movement along the Z-axis between the upper dead point and the lower dead point with sine curve feedrate *F* by Exp. 1. Oscillation override *k* is enabled.



To cancel oscillation motion, specify the following command:

G80 ;

Motion of cancel is as follows.

Bit 0 (OST) of parameter No.25651 is

0: The tool moves to point R and stops.

1: The tool decelerates and stops.

Superimposed control

If oscillation axis is set to master axis and superimposed control is started, it is necessary on the corresponding path to be executing superimposed ahead mode by adjusting superimposed ahead signal OVLN <Gn531.4> to "1". Set superimposed ahead signals OVLN to "1" in each path, if paths of master axis and slave axis are different.

Refer to the following manual for details of the setting method and so on.

"FANUC Series 0*i*-F Plus CONNECTION MANUAL (FUNCTION)" (B-64693EN-1) "Superimposed Control"

Example)

(1) In the case that paths of master axis and slave axis are the same.

```
M210     …Superimposed ahead mode ON
M60      …Starts superimposed control
G81.1    …Starts oscillation motion

G80      …Cancels oscillation motion
M61      …Cancels superimposed control
M211     …Superimposed ahead mode OFF
```

(2) In the case that paths of master axis and slave axis are different.

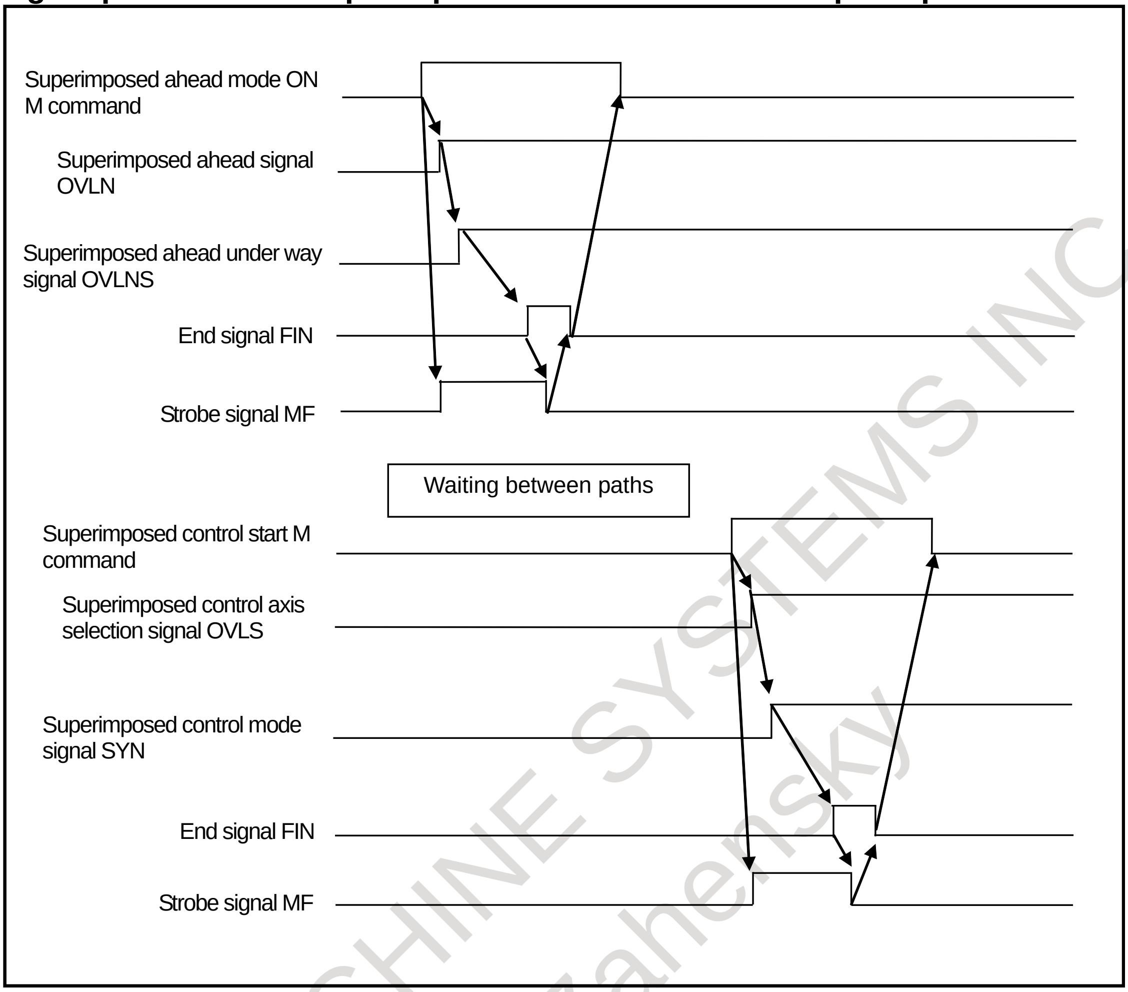
Path of the master axis		Path of the slave axis	
M100P12	…Waiting	M100P12	…Waiting
M210	…Superimposed ahead mode ON	M212	…Superimposed ahead mode ON
		M60	…Starts superimposed control
M101P12	…Waiting	M101P12	…Waiting
G81.1	…Starts oscillation motion		
G80	…Cancels oscillation motion		
M103P12	…Waiting	M103P12	…Waiting
		M61	…Cancels superimposed control
M211	…Superimposed ahead mode OFF	M213	…Superimposed ahead mode OFF
M104P12	…Waiting	M104P12	…Waiting

NOTE

1 To switch superimposed ahead signal ON/OFF, it is necessary to stop all axes (except for PMC axis) of path where this signal is switched.
2 Switch ON/OFF the superimposed ahead mode after the superimposed control has been canceled in slave path.
3 The following functions cannot be commanded in the path to which the superimposed ahead mode is effective.
   - Reference return in Cs contouring control (G00, G28)
   - Skip function (G31)
   - Automatic tool length measurement / Automatic tool offset function
   - Automatic reference return operation of low-speed type (G28)
   - High speed program check function

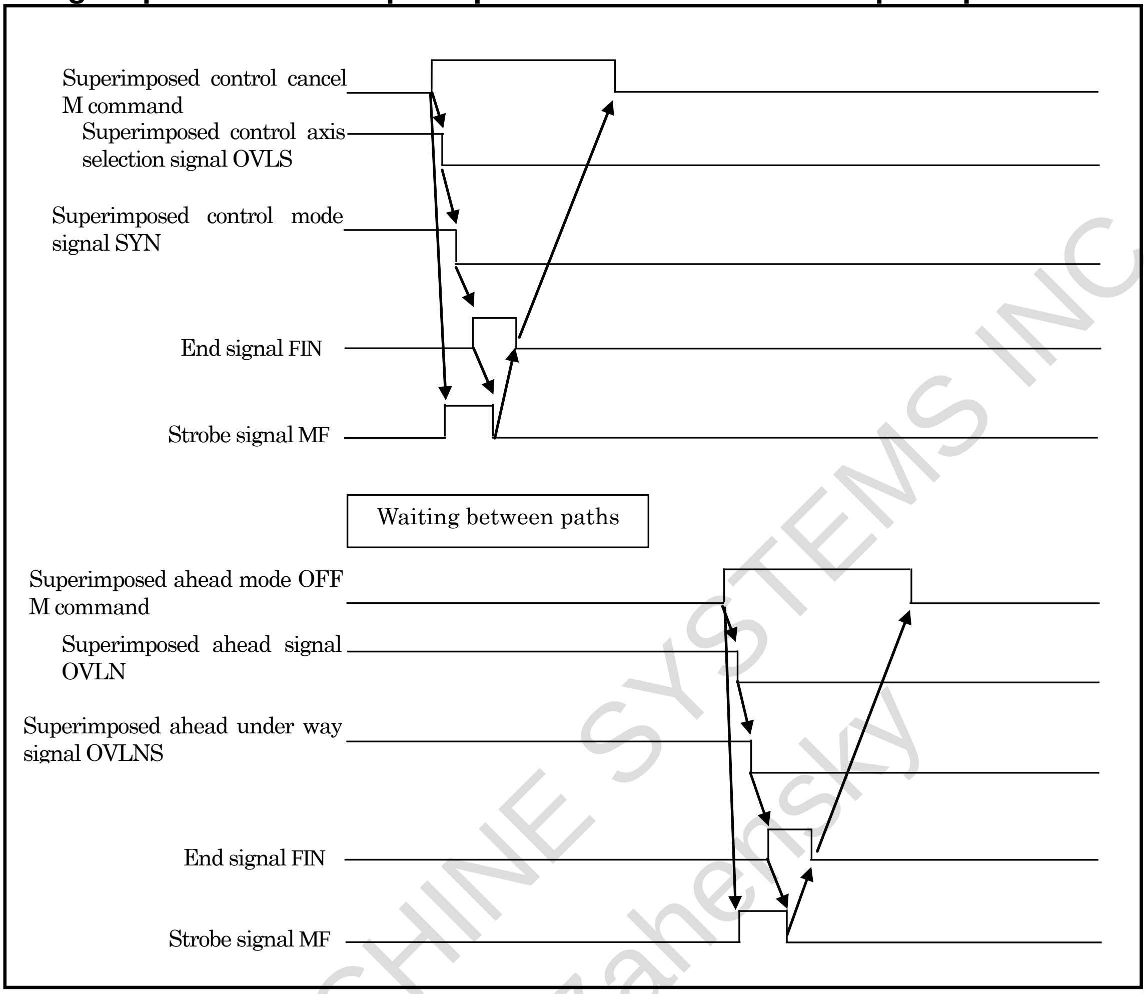
   These functions can be commanded, when superimposed control and superimposed ahead mode are off.
4 The M code for superimposed ahead mode must be assigned to M codes preventing buffering. (Parameter Nos. 3411 to 3432)
5 Oscillation axis cannot be set to slave axis. If oscillation axis is set to slave axis and oscillation motion is started, an alarm PS5050 is issued.

Starting sequence of the superimposed ahead mode and superimposed control.



1. When the M code for enabling superimposed ahead mode is specified and superimposed ahead signal OVLN is set to "1", Superimposed ahead under way signal OVLNS becomes "1".
2. When superimposed ahead under way signal OVLNS becomes "1", operate the end signal FIN in order to complete the procedure.
3. Command the waiting M code in both paths of master axis and slave axis. (If paths of master axis and slave axis belong to the same path, the M code is omitted.)
4. When the M code for enabling superimposed control is specified and superimposed control axis select signal OVLS is set to "1", superimposed control starts. And the superimposed control mode signal SYN becomes "1".
5. When superimposed control mode signal SYN becomes "1", operate the end signal FIN in order to complete the procedure.

Canceling sequence of the superimposed ahead mode and superimposed control.



1. When the M code for canceling the superimposed control is specified, and superimposed control axis select signal OVLS is set to "0", superimposed control will be canceled (superimposed control under way signal SYN turn to “0”).
2. When superimposed control mode signal SYN becomes "0", operate the end signal FIN in order to complete the procedure.
3. Command the waiting M code in both paths of master axis and slave axis. (If paths of master axis and slave axis belong the same path, the M code is omitted.)
4. When the M code for disabling superimposed ahead mode is specified and superimposed ahead signal OVLN is set to "0", superimposed ahead mode is canceled (advanced superimposition mode signal OVLNS turn to “0”).
5. When superimposed ahead under way signal OVLNS becomes to "0", operate the end signal FIN in order to complete procedure.

Signal

Oscillation start signal CHPST <Gn051.6>

[Classification] Input signal
[Function] Starts and stops oscillation.
[Operation] Setting this signal to "1" starts oscillation.
Setting this signal to "0" during oscillation motion causes oscillation to be stopped.

NOTE
1 If an oscillation operation due to the oscillation start signal CHPST has been cancelled by an operation or command that causes it to stop, return the signal CHPST to "0" and then set it "1" again.
2 This signal is not enabled until the completion of manual reference position return.
3 The M codes which are used to set the oscillation start signal CHPST <Gn051.6> to "1" or "0" should be specified as M code preventing buffering (parameter Nos. 3411 to 3432).

Oscillation hold signal *CHLD <Gn051.7>

[Classification] Input signal
[Function] Suspends oscillation.
[Operation] Once this signal has been set to "0", the tool is moved from the current position to point R, thus oscillation is suspended. Setting this signal to "1" while oscillation is suspended causes oscillation to be restarted.

Oscillation feedrate override signals *CHP1 to *CHP8 <Gn051.0 to 3>

[Classification] Input signal
[Function] Overrides the oscillation feedrate.
[Operation] The actual base feedrate during oscillation becomes the specified base feedrate multiplied by the override value specified with this signal. The change amount by the override is affected the parameter No.25653 (Acceleration rate of starting or cancellation oscillation motion). The following table lists the correspondence between the signal states and the override value:

*CHP8	*CHP4	*CHP2	*CHP1	Override value
0	0	0	0	150%
0	0	0	1	140%
0	0	1	0	130%
0	0	1	1	120%
0	1	0	0	110%
0	1	0	1	100%
0	1	1	0	90%
0	1	1	1	80%
1	0	0	0	70%
1	0	0	1	60%
1	0	1	0	50%
1	0	1	1	40%
1	1	0	0	30%
1	1	0	1	20%
1	1	1	0	10%
1	1	1	1	0%

NOTE
According to the range of oscillation motion and the oscillation base federate (Parameter No.8374), shock might occur when override is changed.

Superimposed ahead signal OVLN <Gn531.4>

[Classification] Input signal
[Function] Start superimposed ahead mode.
[Operation] When this signal is "1", the control device operates as follows.

- The corresponding path enters the superimposed ahead state.

To switch superimposed ahead signal to "1" or "0", it is necessary to stop all axes (except for PMC axis) of path where this signal is switched. When axis moves, alarm DS0071 is issued.

Oscillation-in-progress signal CHPMD <Fn039.2>

[Classification] Output signal
[Function] Notify the state of oscillation
[Operation] This signal is set to "1" in the following cases:

- Oscillation start signal CHPST is changed from "0" to "1" to start oscillation.
- Oscillation is started by G81.1 command.

This signal is set to "0" in the following cases:

- Oscillation start signal CHPST is changed from "1" to "0" to stop oscillation.
- Oscillation is terminated by a reset.
- Oscillation is cancelled by G80 command.
- Oscillation is terminated when an alarm is issued.
- Oscillation is terminated by emergency stop.

Oscillation cycle signal CHPCYL <Fn039.3>

[Classification] Output signal
[Function] Notify the state of an oscillation cycle being performed between the upper and lower dead points.
[Operation] This signal is set to "1" in the following case:

- Oscillation cycle is started and sign curve federate control is used between the upper and lower dead points.

This signal is set to "0" in the following cases:

- Once oscillation has been stopped
- When the tool is stopped at the upper or lower dead point
- Oscillation hold signal *CHLD is set to "0"

Superimposed ahead under way signal OVLNS <Fn545.1>

[Classification] Output signal
[Function] Indicates that superimposed ahead mode is executing.
[Operation] The signal becomes "1" when:

- The corresponding path is executing superimposed ahead mode.

The signal becomes "0" when:

- The corresponding path is not executing superimposed ahead mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn051	*CHLD	CHPST			*CHP8	*CHP4	*CHP2	*CHP1
Gn531				OVLN				
Fn039					CHPCYL	CHPMD		
Fn545							OVLNS	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8360	CHF					CVC		ROV

[Input type] Setting input
[Data type] Bit path

#0 ROV As rapid traverse override for a section from the oscillation start point to point R:
0: Oscillation override is used.
1: Rapid traverse override is used.

#2 CVC The feedrate along the oscillation axis is changed:
0: At the upper or lower dead point immediately after the feedrate change command is issued.
1: At the upper dead point immediately after the feedrate change command is issued.

#7 CHF On the oscillation screen, the oscillation base feedrate:
0: Can be set.
1: Cannot be set.

8370	Oscillation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes
This parameter sets which servo axis the oscillation axis corresponds to.

8371	Oscillation reference point (point R)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the oscillation axis
[Valid data range] 9 digits of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
The data set in this parameter is absolute coordinates.

8372	Oscillation upper dead point

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the oscillation axis
[Valid data range] 9 digits of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
The data set in this parameter is absolute coordinates.

8373	Oscillation lower dead point

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the oscillation axis

[Valid data range] 9 digits of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
The data set in this parameter is absolute coordinates.

8374	Oscillation base feedrate

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (input unit)
[Min. unit of data] Depend on the increment system of the oscillation axis
[Valid data range] Refer to standard parameter setting table (C).
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the oscillation feedrate.

8375	Maximum oscillation feedrate

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to standard parameter setting table (C).
(When the increment system is IS-B, 0.0 to +999000.0)
The oscillation feedrate is clamped at this parameter setting. The maximum feedrate must be set for the oscillation axis. If this parameter is set to 0, no movement is made for oscillation.

> **NOTE**
> Please set a value that is smaller than the normal rapid traverse rate (parameter No. 1420) to Maximum oscillation feedrate.

	#7	#6	#5	#4	#3	#2	#1	#0
25650								SSO

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 SSO High precision oscillation function is
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
25651					HST	SGS	FFS	OST

[Input type] Parameter input
[Data type] Bit path

#0 OST During oscillation motion, if G80 command or reset is specified,
0: Oscillation axis moves to point R and stops. (Standard specification)
1: Oscillation axis decelerates and stops.

#1 FFS When oscillation motion is started by G81.1 command,

0: Oscillation motion is begun, after the oscillation axis passes point R and the center point between upper and lower dead points are passed. (Standard specification)

1: Movements to point R and to the center point between upper and lower dead points is omitted, and oscillation motion can be immediately started.

If this parameter is set to 1, the first dead point is as follows.

State before G81.1 command	The first dead point
First command of oscillation motion after the power is turned on or in the state of reset.	Lower dead point
In the state that command of cancel was specified on the way to lower dead point at last oscillation motion.	
In the state that command of cancel was specified on the way to upper dead point at last oscillation motion.	Upper dead point

NOTE
If this parameter is effective, bit 0 (OST) of parameter No. 25651 need be set to 1. Then the tool decelerates and stops when the oscillation motion is canceled.

#2 SGS If oscillation motion is canceled by turning oscillation start signal CHPST <Gn051.6> from “1” to “0”,

0: Oscillation axis moves to point R and stops. (Standard specification)

1: Oscillation axis decelerates and stops.

If the oscillation axis decelerates and stops by oscillation start signal CHPST, use each axis workpiece coordinate system preset signal WPRST1-WPRST8<Gn358> together.
Refer to the following manual for details of the setting method and so on.

"FANUC Series 0*i*-F Plus CONNECTION MANUAL (FUNCTION)" (B-64693EN-1) "Each Axis Workpiece Coordinate System Preset Signals"

NOTE
If this parameter is effective, bit 0 (OST) of parameter No. 25651 need be set to 1.

#3 HST During oscillation motion, if oscillation hold signal *CHLD <Gn051.7> is set to"0" from "1",

0: Oscillation axis moves to point R and suspends. (Standard specification)

1: Oscillation axis decelerates and suspends.

NOTE
If this parameter is effective, bits 0 (OST) and 2 (SGS) of parameter No. 25651 need be set to 1.

25652	Maximum allowable acceleration rate of oscillation motion

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Valid data range] Refer to standard parameter setting table (D). (Operating range: 0.0 to +100000.0)

Set a maximum allowable acceleration rate for oscillation axis during oscillation motion (with sine curve feedrate).

If a value greater than maximum value (100000.0) is set, the value is clamped to maximum value (100000.0).
If 0 is set, maximum value (100000.0) is assumed to be set.

25653	Acceleration rate of starting or cancellation oscillation motion

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Valid data range] Refer to standard parameter setting table (D). (Operating range: 0.0 to +100000.0)

It is the linear acceleration rate of starting motion (from point R to center point between upper and lower dead points) or canceling motion.
Moreover, when override is changed during oscillation motion, the acceleration/deceleration is set by this parameter.
If a value greater than maximum value (100000.0) is set, the value is clamped to maximum value (100000.0).
If 0 is set, maximum value (100000.0) is assumed to be set.

	#7	#6	#5	#4	#3	#2	#1	#0
25655							CST	

[Input type] Parameter input
[Data type] Bit path

#1 CST When Oscillation start signal CHPST <Gn051.6> is changed to “0” or when the alarm (except BG and OH) is issued while moving to point R from the start of oscillation by Oscillation start signal CHPST:
0: Cancel of oscillation mode is executed after reaching point R.
1: Cancel of oscillation mode is executed immediately.

When reset operation is executed while moving to point R, cancel of oscillation mode is executed immediately regardless of the setting of this parameter.

> **NOTE**
> In the following case, this parameter does not affect the motion.
> 1 Start of oscillation by G81.1 command
> 2 Oscillation motion after reaching point R

Alarm and message

Number	Message	Description
PS5050	ILL-COMMAND IN G81.1 MODE	- During oscillation, a move command has been issued for the oscillation axis. - Oscillation is specified for Cs contour control axis. - Machine lock for the axis while oscillation motion is executed. - Oscillation is specified for the slave axis of superimposed control.

Reference item

Manual name	Item name
FANUC AC SERVO MOTOR α*i* series PARAMETER MANUAL (B-65270EN)	Look-ahead feed forward function
FANUC Series 0*i*-F Plus CONNECTION MANUAL (FUNCTION) (B-64693EN-1)	Superimposed Control
	Each Axis Workpiece Coordinate System Preset Signals

1.16 DUAL CONTROL AXES SWITCHING

Overview

This function makes it possible to allocate two control axes to one motor. When the allocated two control axes are in the controlled axes detach, the allocated two control axes can be switched. The reference position is not lost at the change of the control axes, when the absolute position detection is used. This function is useful for using the attachment with different gear ratio by one motor.

Explanation

Two control axes are allocated to one motor. As a result, one motor can have two independent axes setting (parameter, compensation, and custom macro system variables, etc. that related to axis).
Number of the servo axis (parameter No.1023) of these two control axes must set the consecutive values that the odd number and the even number (odd number + 1).The odd number is set to parameter No.1023 for a real axis, and the even number is set to parameter No.1023 for an imaginary axis.
It is necessary for a real axis that the setting (bit 3 (DCHx) of parameter No.2437 is set to 1) which the dual control axes switching function is enabled.
It is necessary for an imaginary axis that the setting (bit 3 (DCHx) of parameter No.2437 and bit 0 (DMYx) of parameter No.2009 are set to 1) which the dual control axes switching function and the serial feedback dummy function are enabled. And set the dummy axis for FSSB setting too.
When both a real axis and an imaginary axis are in the controlled axes detach, the selected axis and the non-selected axis can be switched by dual control axes switching signal SVMWCx <Gn687.x>.
The motor moves according to command for the selected axis.
Feedback from the motor returns to both the selected axis and the non-selected axis. A position display of the selected axis and the non-selected axis (machine position display, absolute position display and relative position display) are updated by the feedback pulses according to the setting of each axis.

Table 1.16 (a) Relationships between dual control axes switching signal and selected axis

	Dual control axes switching signal SVMWCx <Gn687.x>	
	0	1
Selected axis	Real axis	Imaginary axis
Non-selected axis	Imaginary axis	Real axis

[Example]
Real axis (C1) : Parameter No.1023 = 3
Bit 3 (DCHx) of parameter No.2437 = 1
Imaginary axis (C2) : Parameter No.1023 = 4
Bit 0 (DMYx) of parameter No.2009 = 1
Bit 3 (DCHx) of parameter No.2437 = 1

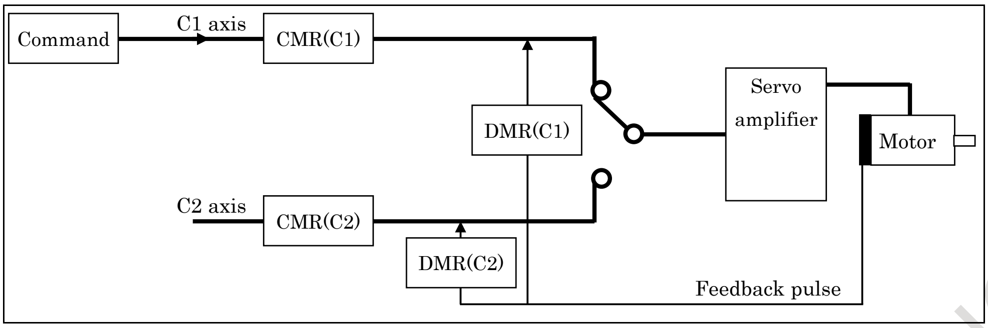


Fig. 1.16 (a) Example of dual control axes switching signal SVMWCx <Gn687.x> = "0"
(selected axis: C1, non-selected axis: C2)

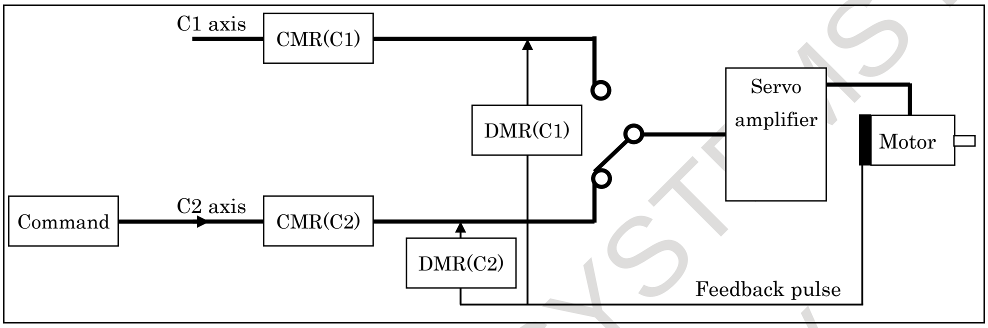


Fig. 1.16 (b) Example of dual control axes switching signal SVMWCx <Gn687.x> = "1"
(selected axis: C2, non-selected axis: C1)

Full closed system

This function can be controlled in the full closed system by using two separate position detectors. Feedback from the separate position detectors returns to each axis when the full closed system is used.
It is necessary for an imaginary axis to set 1 to bit 3 (DCFULLx) of parameter No.2570.

[Example]

Real axis (C1) : Parameter No.1023 = 3
Bit 3 (DCHx) of parameter No.2437 = 1
Parameter No.24096 (C1) = 1

Imaginary axis (C2) : Parameter No.1023 = 4
Bit 0 (DMYx) of parameter No.2009 = 1
Bit 3 (DCHx) of parameter No.2437 = 1
Bit 3 (DCFULLx) of parameter No.2570 = 1
Parameter No.24096 (C2) = 2

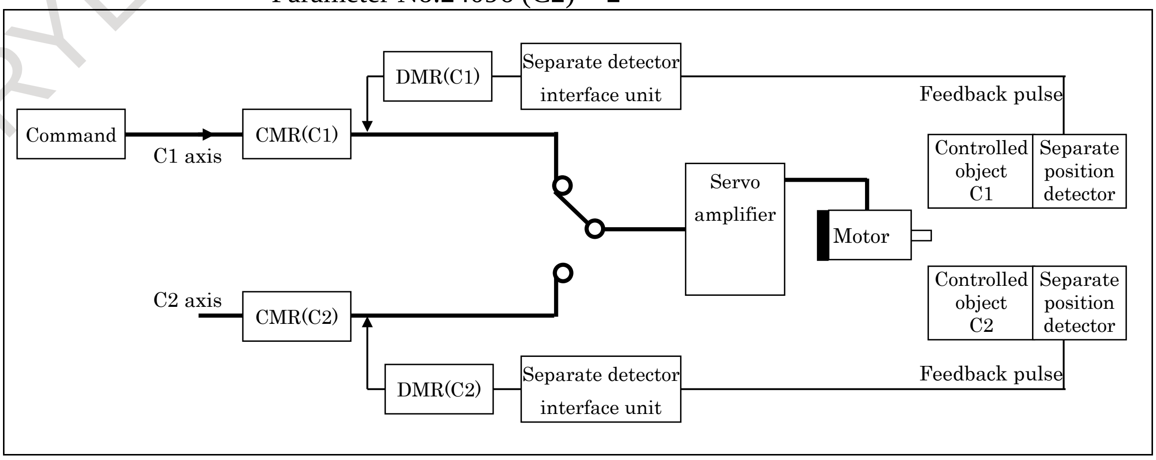


Fig. 1.16 (c) Example of dual control axes switching signal SVMWCx <Gn687.x> = "0" in full closed system
(selected axis: C1, non-selected axis: C2)

> **NOTE**
> 1 A different system (semi closed system, full closed system) cannot be used between a real axis and an imaginary axis.
> 2 When one separate position detector is shared, it is necessary to set 0 to bit 3 (DCFULLx) of parameter No.2570.

[Switch procedure of dual control axes switching signal]

To switch the dual control axes switching signal from “0” to “1”, switch the signal according to the following procedure.

(1) Set the controlled axis detach signals DTCHx <Gn124.x> for a real axis and an imaginary axis to “1”, using the M code as a trigger.
(2) Set the dual control axes switching signal SVMWCx <Gn687.x> to “1”, when the controlled axis detach status signals MDTCHx <Fn110.x> for a real axis and an imaginary axis are “1”.
(3) Set the controlled axis detach signals DTCHx <Gn124.x> for a real axis and an imaginary axis to “0”, when the dual control axes status signal SVMWSx <Fn687.x> is “1”.
(4) Perform the FIN processing for the M code.

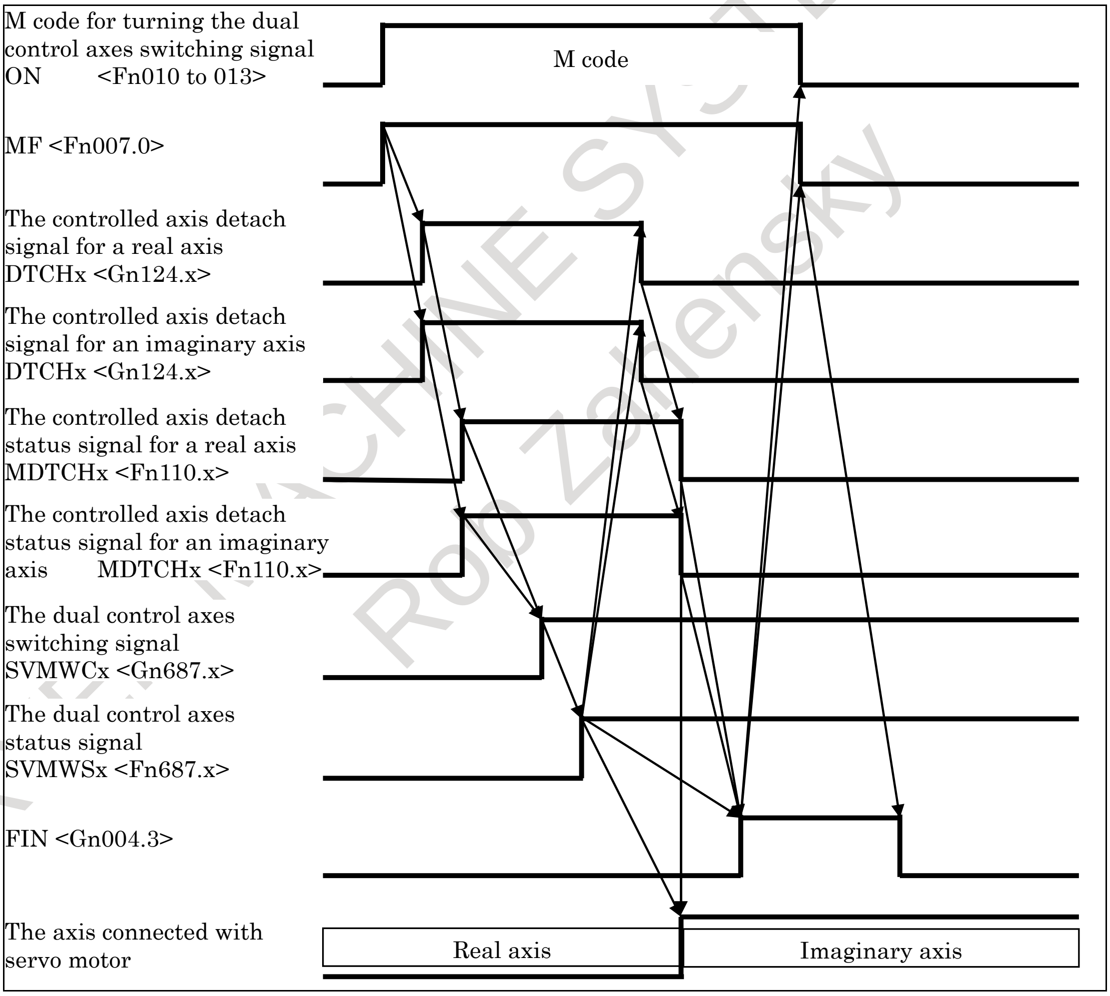


Fig. 1.16 (d) Timing chart (The dual control axes switching signal SVMWCx <Gn687.x> “0” → “1”)

To switch when the dual control axes switching signal from “1” to “0”, switch the signal according to the following procedure.

(1) Set the controlled axis detach signals DTCHx <Gn124.x> for a real axis and an imaginary axis to “1”, using the M code as a trigger.
(2) Set the dual control axes switching signal SVMWCx <Gn687.x> to “0”, if the controlled axis detach status signals MDTCHx <Fn110.x> for a real axis and an imaginary axis are “1”.

(3) Set the controlled axis detach signals DTCHx <Gn124.x> for a real axis and an imaginary axis to "0", if the dual control axes status signal SVMWSx <Fn687.x> is "0".
(4) Perform the FIN processing for the M code.

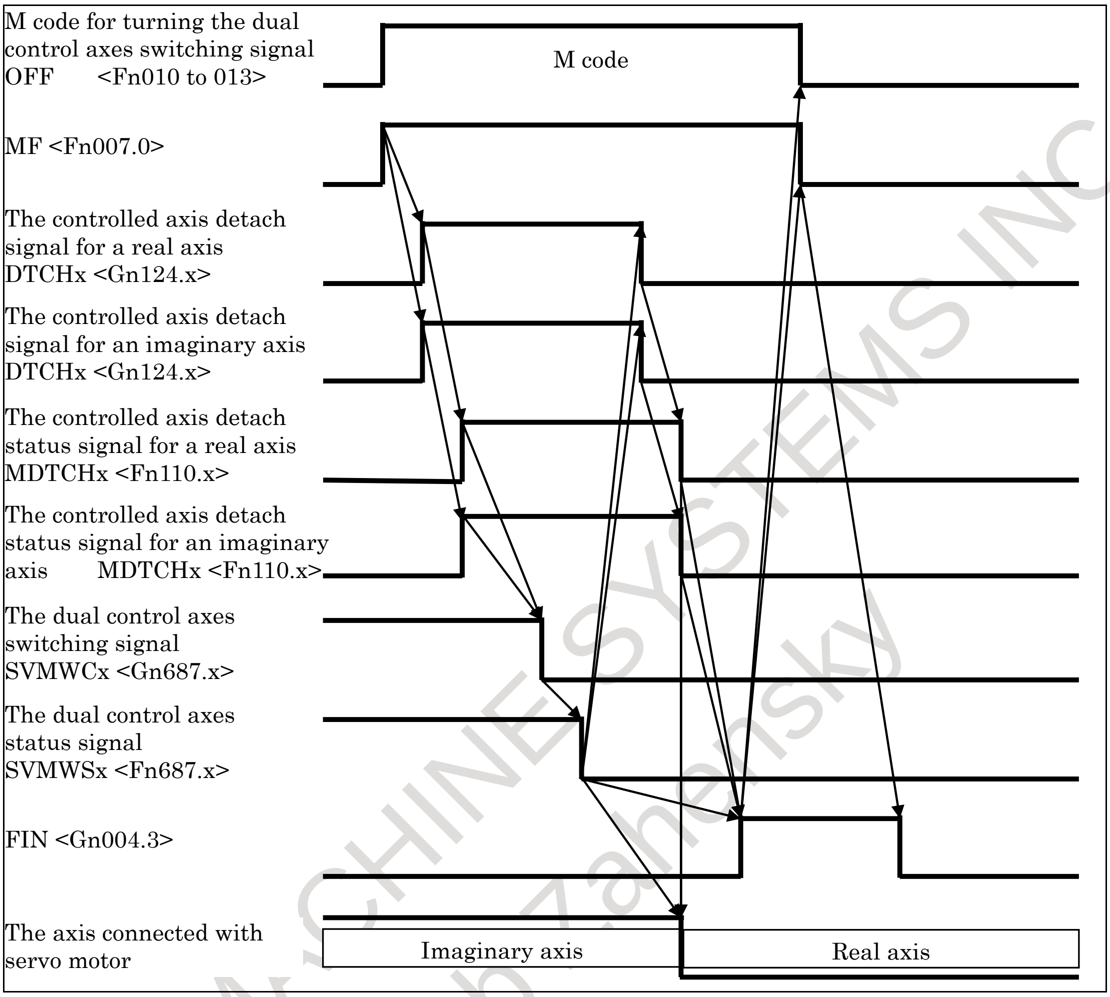


Fig. 1.16 (e) Timing chart (The dual control axes switching signal SVMWCx <Gn687.x> "1" → "0")

Use with absolute position detection

To set the zero point of an absolute position detector, perform the manual reference position return at the selected axis. Perform the manual reference position return for a real axis and an imaginary axis as the selected axis.
If the zero point of an absolute position detector is set, the manual reference position return is unnecessary when the dual control axes switching signal SVMWCx <Gn687.x> is switched.

Without absolute position detection

When the dual control axes switching signal SVMWCx <Gn687.x> is switched, the reference position is lost. The manual reference position return must be performed by the selected axis before the move command. If specifying the move command without reference position return, alarm PS0224 "ZERO RETURN NOT FINISHED" is issued.

Restrictions

Dual control axes switching cannot be used together with the following functions.

- Dual Check Safety
- Angular axis control
- Tandem control
- Dual position feedback

- Semi-Full error monitoring

The dual control axes switching cannot be used for multiple winding motor.

Disable the following functions, before the dual control axes switching signal SVMWCx <Gn687.x> is switched.

- Synchronous control and composite control
- Superimposed control
- Axis synchronous control
- Flexible synchronization control
- Electronic gear box
- Axis control by PMC
- Spindle control with servo motor

NOTE

1 To disable the display of non-selected axis, delete the current position indication for the non-selected axis by using bit 0 (NDPx) of parameter No.3115.
2 The setting of bit 0 (ROTx) and 1 (ROSx) of parameter No.1006 should be same setting for both a real axis and an imaginary axis. If the setting is different, the machine coordinates might not be correctly updated at dual control axes switching signal SVMWCx <Gn687.x> switching.
3 If the controlled axis detach status signals MDTCHx <Fn110.x> for a real axis and an imaginary axis are not “1” while dual control axes switching signal SVMWCx <Gn687.x> is switched, alarm SV0401 “IMPROPER V_READY OFF” is issued.
4 In automatic operation, set the M code for dual control axes switching signal on or off to the M code preventing buffering. (Parameters Nos.3411 to 3432)
5 An alarm occurs at the time of a mechanical adjustment, so on and off of dual control axes switching signal perform a time chart on Fig. 1.16 (d) and Fig. 1.16 (e) manually.
6 If the dual control axes switching is executed while excitation of the motor is on, alarm SV0417 “ILL DGTL SERVO PARAMETER” is issued. (Detail number of diagnosis data No.352: 4379)
7 It is necessary to change bit 1 to 4 of parameter No.2013 and bit 1 to 4 of parameter No.2014 for an axis of the dual control axes switching according to the motor model. If the standard parameters for the motor model are the same as the values described in the before change column in the below table, change to the values described in the after change column.

No.2013 #4,#3,#2#1 No.2014 #4,#3,#2,#1		
Before change		After change
1, 0, 0, 1	→	1, 0, 1, 0
0, 1, 1, 1	→	1, 0, 0, 0
0, 1, 0, 1	→	0, 1, 1, 0
0, 0, 1, 1	→	0, 1, 0, 0
0, 0, 0, 1	→	0, 0, 1, 0

8 To get the frequency response using SERVO GUIDE, because the serial feedback dummy function (bit 0 (DMYx) of parameter No.2009) is effective to an imaginary axis, it is not possible to measure. To get the frequency response with imaginary axis, temporarily invalidate the serial feedback dummy function (bit 0 (DMYx) of parameter No.2009) for an imaginary axis after power on and measure it. After completing the measurement, reset to effective the serial feedback dummy function (bit 0 (DMYx) of parameter No.2009 = 1).
9 In actual speed display, a synthetic speed of real axis and imaginary axis is displayed. In order to exclude imaginary axis speed from the actual speed display, set actual speed display axis selection signals *ACTF1 to *ACTF8 <Gn580> to “1” about the imaginary axis.

Parameter

1023	Number of the servo axis for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 80

This parameter associates each control axis with a specific servo axis. Specify values 1+8n, 2+8n, 3+8n, 4+8n, 5+8n, and 6+8n (n = 0, 1, 2, …, 9) like 1, 2, 3, 4, 5, …, 77, and 78.
The control axis number is the order number that is used for setting the axis-type parameters or axis-type machine signals
For dual control axes switching, two axes need to be specified as one pair. So, make a setting as described below.
For a real axis, set an odd (1, 3, 5, 9, ...) servo axis number. For an imaginary axis to be paired, set a value obtained by adding 1 to the value set for the real axis.

	#7	#6	#5	#4	#3	#2	#1	#0
2009								DMYx

[Input type] Parameter input
[Data type] Bit axis

#0 DMYx The serial feedback dummy function is:
0: Not used.
1: Used.
Set this parameter for the imaginary axis.

> **NOTE**
> The setting of this parameter becomes valid after the power is turned off then back on.

	#7	#6	#5	#4	#3	#2	#1	#0
2437					DCHx			

[Input type] Parameter input
[Data type] Bit axis

#3 DCHx The dual control axes switching function is:
0: Disabled.
1: Enabled.
Set this parameter to both axes of the real axis and the imaginary axis.

	#7	#6	#5	#4	#3	#2	#1	#0
2570					DCFULLx			

[Input type] Parameter input
[Data type] Bit axis

#3 DCFULLx In the dual control axes switching function, one scale is:
0: Shared with the real axis and the imaginary axis.
1: Not shared with the real axis and the imaginary axis.
Set this parameter only for the imaginary axis.

NOTE
In the case of having an individual scale with the real axis and imaginary axis, set this bit to 1.

24096	**Connector number for the 1st or 9th separate detector interface unit**
24097	**Connector number for the 2nd or 10th separate detector interface unit**
24098	**Connector number for the 3rd or 11th separate detector interface unit**
24099	**Connector number for the 4th or 12th separate detector interface unit**
24100	**Connector number for the 5th separate detector interface unit**
24101	**Connector number for the 6th separate detector interface unit**
24102	**Connector number for the 7th separate detector interface unit**
24103	**Connector number for the 8th separate detector interface unit**

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 8

Set a connector number for the connector to which a separate detector interface unit is attached if the separate detector interface unit is to be used. The following table lists the necessary settings. Be sure to specify 0 for connectors not in use.

Correspondence between connectors and connector numbers

Connector	Connector number
JF101	1
JF102	2
JF103	3
JF104	4
JF105	5
JF106	6
JF107	7
JF108	8

(Setting example)

Controlled axis	Connector to which each separate detector interface unit is attached				Parameter setting			
	1st connector	2nd connector	5th connector	6th connector	No. 24096	No. 24097	No. 24100	No. 24101
X1	JF101	–	–	–	1	0	0	0
Y1	–	JF102	–	–	0	2	0	0
Z1	–	–	JF102	–	0	0	2	0
X2	–	JF101	–	–	0	1	0	0
Y2	–	–	–	JF101	0	0	0	1
Z2	–	–	–	–	0	0	0	0
A1	–	–	JF101	–	0	0	1	0
B1	–	–	–	JF102	0	0	0	2
C1	–	JF104	–	–	0	4	0	0
A2	JF102	–	–	–	2	0	0	0
B2	–	JF103	–	–	0	3	0	0
C2	–	–	–	JF103	0	0	0	3

NOTE

1. Specify these parameters when separate detector interface units are used.
2. Parameters Nos. 24096 to 24103 are specified automatically when data is entered on the FSSB setting screen if the FSSB setting mode in use is the automatic setting mode (bit 0 (FMD) of parameter No. 1902 = 0). If the manual setting 2 mode (bit 0 (FMD) of parameter No. 1902) = 1), specify the parameters directly.

Alarm and message

Number	Message	Description
PS0224	ZERO RETURN NOT FINISHED	A reference return has not been performed before the start of automatic operation. (Only when bit 0 (ZRNx) of parameter No.1005 is 0) Perform a reference position return.
SV0401	IMPROPER V_READY OFF	Although the ready signal (PRDY) of the position control was ON, the ready signal (VRDY) of the velocity control was OFF.
SV0417	ILL DGTL SERVO PARAMETER	A digital servo parameter setting is incorrect.

Signal

Actual speed display axis selection signals *ACTF1 to *ACTF8 <Gn580>

[Classification] Input signal

[Function] These signals specify whether the movement speed of each axis is added to the display of actual speed.

[Operation] The movement speed of the axis for which 1 is set is not added to the display of actual speed.

Dual control axes switching signal SVMWC1 to SVMWC8 <Gn687>

[Classification] Input signal

[Function] These signals switch the selected axes in dual control axes.

[Operation] When this signal is set to “0”, the real axis becomes selected axis. And the imaginary axis becomes non-selected axis.
When this signal is set to “1”, the imaginary axis becomes selected axis. And the real axis becomes non-selected axis.

Dual control axes status signal SVMWS1 to SVMWS8 <Fn687>

[Classification] Output signal

[Function] This signal notifies the status of the dual control axes.

[Operation] This signal becomes “0” when:
- The real axis is selected axis, and the imaginary axis is non-selected axis.

This signal becomes “1” when:
- The imaginary axis is selected axis, and the real axis is non-selected axis.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn580	*ACTF8	*ACTF7	*ACTF6	*ACTF5	*ACTF4	*ACTF3	*ACTF2	*ACTF1

	#7	#6	#5	#4	#3	#2	#1	#0
Gn687	SVMWC8	SVMWC7	SVMWC6	SVMWC5	SVMWC4	SVMWC3	SVMWC2	SVMWC1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn687	SVMWS8	SVMWS7	SVMWS6	SVMWS5	SVMWS4	SVMWS3	SVMWS2	SVMWS1

NOTE
The signal (Gn687, Fn687) for the real axis can be used. The signal for the imaginary axis cannot be used.

1.17 VELOCITY FEED FORWARD COMPENSATION FUNCTION

Overview

Velocity feed forward coefficient can be adjusted according to the machine coordinate value of the moving axis specified by parameter No.11821.

Explanation

Velocity feed forward coefficient of the compensation axis can be adjusted according to the machine coordinate value of the moving axis specified by parameter No.11821 with this function.

Velocity feed forward coefficient of the compensation axis is determined by the following formula.

$$VFF = \frac{K}{100} VFF_0$$

VFF : Velocity feed forward coefficient
VFF_0 : Velocity feed forward coefficient parameter (No.2069, 2145)
K : Compensation coefficient

Compensation coefficient of the compensation axis is set with the following parameters.

Table 1.17 (a) Compensation coefficient setting parameter

	Parameter
Machine coordinate of moving axis	No.14270 to No.14279
Compensation coefficient of compensation axis	No.14280 to No.14289

Example 1)
When setting the parameter in Table 1.17 (b) below, compensation coefficients of the compensation axis are as shown in Fig. 1.17 (a).

Table 1.17 (b) Setting example 1

Parameter	Setting value	Parameter	Setting value
No.14270	-3000.0	No.14280	100
No.14271	-2500.0	No.14281	150
No.14272	-2000.0	No.14282	100
No.14273	-800.0	No.14283	100
No.14274	-500.0	No.14284	50
No.14275	500.0	No.14285	50
No.14276	800.0	No.14286	100
No.14277	2000.0	No.14287	100
No.14278	2500.0	No.14288	150
No.14279	3000.0	No.14289	100

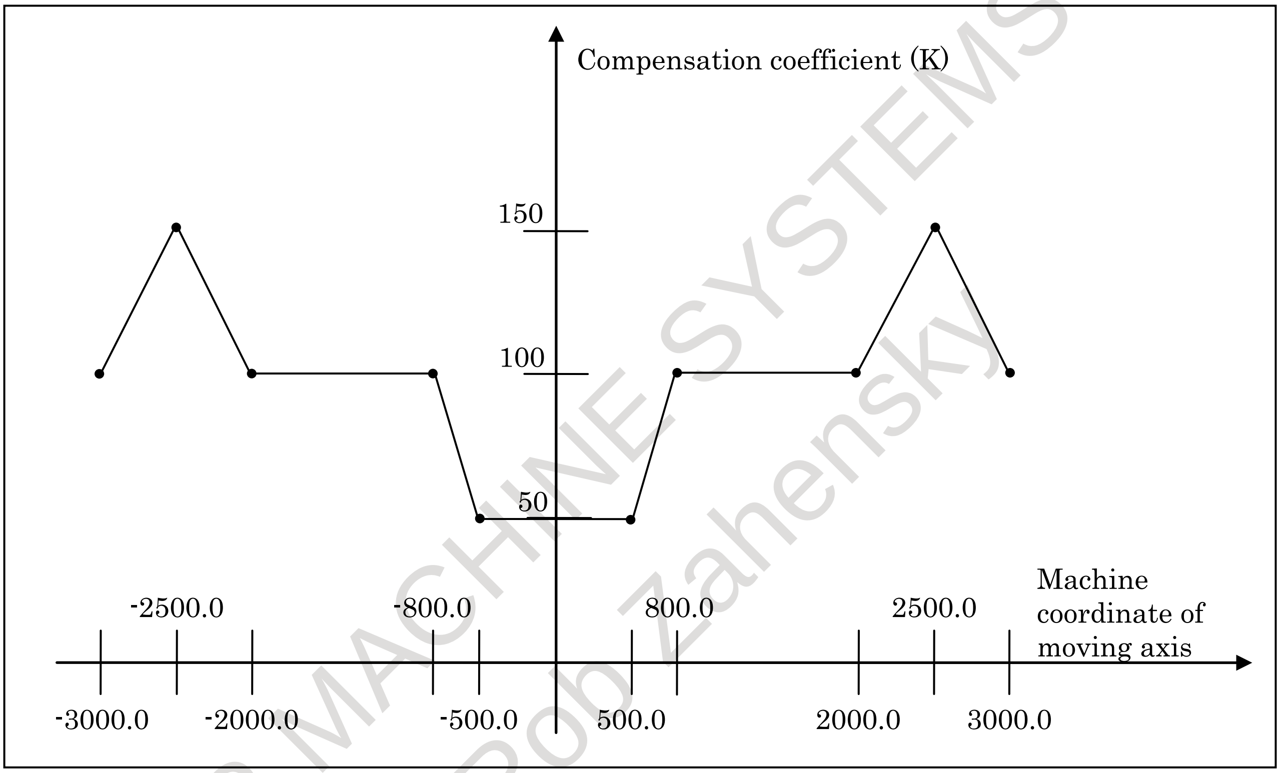


Fig. 1.17 (a) Compensation coefficient (example 1)

Example 2)
When setting the parameter in Table 1.17 (c) below, compensation coefficients of the compensation axis are as shown in Fig. 1.17 (b) .

Table 1.17 (c) Setting example 2

Parameter	Setting value	Parameter	Setting value
No.14270	-5.0	No.14280	50
No.14271	1000.0	No.14281	150
No.14272	0.0	No.14282	0
No.14273	0.0	No.14283	0
No.14274	0.0	No.14284	0
No.14275	0.0	No.14285	0
No.14276	0.0	No.14286	0
No.14277	0.0	No.14287	0
No.14278	0.0	No.14288	0
No.14279	0.0	No.14289	0

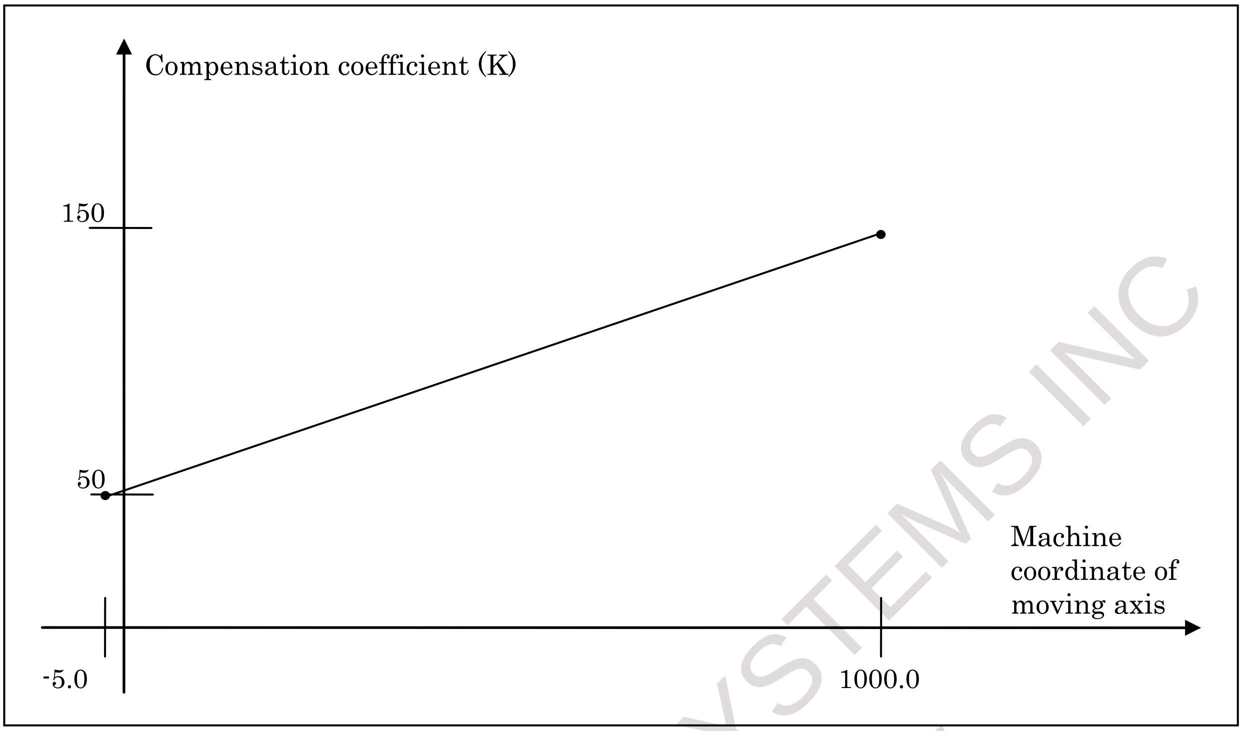


Fig. 1.17 (b) Compensation coefficient (example 2)

> **NOTE**
> 1 This function is enabled after reference position return of moving axis and compensation axis are completed.
> 2 Set the parameter No.14270 to No.14279 setting so that the set value of the small parameter number becomes smaller. (No.14270 setting value < No.14271 setting value < … < No.14279 setting value) When the magnitude relation of the setting value is not established, subsequent parameter setting becomes invalid. In the case of Table 1.17 (c), since the setting value of No.14271 is larger than the setting value of No.14272, parameter setting after No.14272 is invalid.
> 3 This function is invalid if the machine coordinate value of moving axis is out of the range of parameter No.14270 to No.14279 set value.

Limitation

- Moving axis and compensation axis must be assigned to be the same path.
- When the axis used this function, the following function cannot be used.
  - Cs contouring control
  - Spindle control with servo motor
  - Temporary absolute coordinate setting

Parameter

2069	Velocity feedforward coefficient (VFFLT)

[Input type] Parameter input
[Data type] Word axis
[Unit of data] %
[Recommendation] 50 to 200

2145	Velocity feedforward coefficient for cutting feed

[Input type] Parameter input

[Data type] Word axis
[Unit of data] %
[Recommendation] 50 to 200

11821	Controlled axis number of moving axis

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0, 1 to 24
Specify controlled axis number.
When 0 is set, this function is invalid.

> **NOTE**
> 1 When this parameter is set, the power must be turned off before operation is continued.
> 2 Set this parameter on compensation axis.

14270	Machine coordinate 1 of moving axis
14271	Machine coordinate 2 of moving axis
14272	Machine coordinate 3 of moving axis
14273	Machine coordinate 4 of moving axis
14274	Machine coordinate 5 of moving axis
14275	Machine coordinate 6 of moving axis
14276	Machine coordinate 7 of moving axis
14277	Machine coordinate 8 of moving axis
14278	Machine coordinate 9 of moving axis
14279	Machine coordinate 10 of moving axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Specify the machine coordinate value of the moving axis.

> **NOTE**
> 1 When this parameter is set, the power must be turned off before operation is continued.
> 2 Set this parameter on compensation axis.

14280	Compensation coefficient 1 of compensation axis

14281	Compensation coefficient 2 of compensation axis
14282	Compensation coefficient 3 of compensation axis
14283	Compensation coefficient 4 of compensation axis
14284	Compensation coefficient 5 of compensation axis
14285	Compensation coefficient 6 of compensation axis
14286	Compensation coefficient 7 of compensation axis
14287	Compensation coefficient 8 of compensation axis
14288	Compensation coefficient 9 of compensation axis
14289	Compensation coefficient 10 of compensation axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] %
[Valid data range] 1 to 1000

Specify the compensation coefficient of compensation axis.

NOTE
1 When this parameter is set, the power must be turned off before operation is continued.
2 Set this parameter on compensation axis.
3 When a value outside the data range is set, 100 is assumed to be set.

2 PREPARATIONS FOR OPERATION

2.1 EMERGENCY STOP

Overview

By pressing the emergency stop button on the machine operation panel, you can immediately stop the movement of the machine in case of emergency.

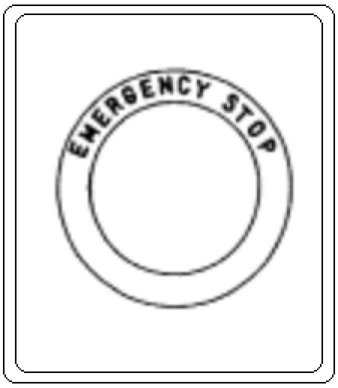


Fig. 2.1 (a) EMERGENCY STOP BUTTON

This button is locked when it is pressed. Although it varies with the machine tool builder, the button can usually be unlocked by twisting it.

Signal

Emergency stop (input)

***ESP<X008.4> (for the first machine group), <X008.0> (for the second machine group), <X008.1> (for the third machine group), <Gn008.4>**

[Classification] Input signal

[Function] Immediately stop the movement of the machine in case of emergency.

[Operation] When the emergency stop signal *ESP turns to “0”, the emergency stop is applied to the machine and the CNC is reset. In general, this signal is controlled by the B contacts of a pushbutton switch. The emergency stop signal turns the servo ready signal SA to “0”.
In this CNC, it is the basic specification that overtravel detection is performed by the stored stroke check function. A limit switch for normal overtravel detection is not needed. To prevent the machine from moving beyond the software limit through servo feedback error, always install a stroke end limit switch (shown in Fig. 2.1 (b) as follows).

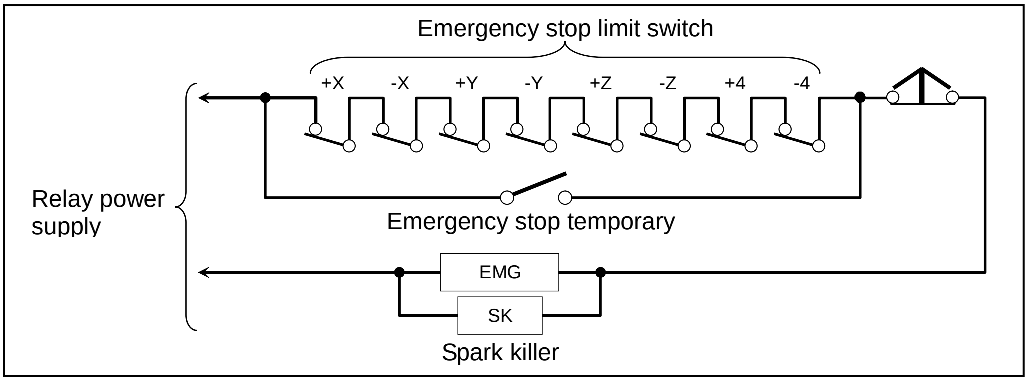
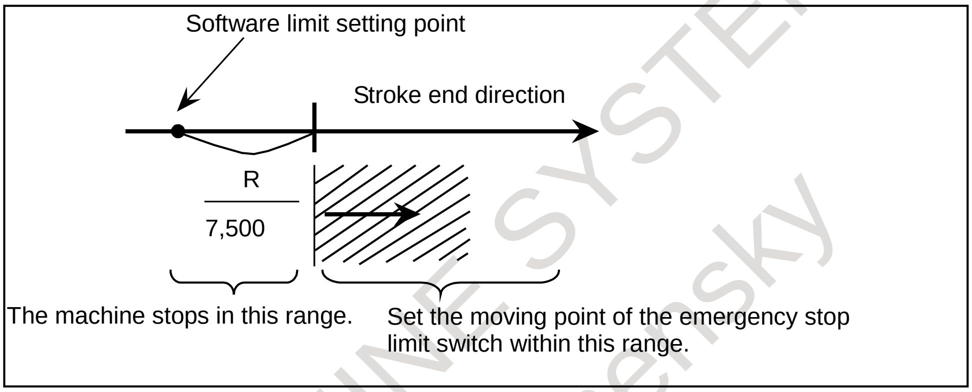


Fig. 2.1 (b) Connection of emergency stop limit switch

The distance from the position where the dynamic brake is applied to that where the tool stops moving is given in the "FANUC AC SERVO MOTOR α*i*-B/α*i* series DESCRIPTIONS (B-65262EN)."

> ⚠ **WARNING**
> About the software limit setting point and operating point of limit switch for emergency stop
> The stop point by the software limit goes beyond the setting point by as much as the following distance.
> The actual stopping point may exceed the position set by parameters Nos.1320 and 1321 by as much as R/7500 (mm).
> R/7,500 (mm)
> R: Rapid traverse rate (mm/min)
> Set the limit switch for emergency stop including the allowance for the above value.



Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X008				*ESP			(*ESP)	(*ESP)
Gn008				*ESP				

Reference item

Manual name	Item name
FANUC AC SERVO MOTOR α*i* series DESCRIPTIONS (B-65262EN)	Servo motor selection

2.2 CNC READY SIGNALS

Overview

When the CNC is turned on and becomes ready for operation, the CNC ready signal is set to 1.

Signal

CNC Ready Signal MA<Fn001.7>

[Classification] Output signal

[Function] The CNC ready signal indicates that the CNC is ready.

[Output cond.] When the CNC is turned on and becomes ready for operation, the signal is set to “1”. Normally, it takes several seconds to establish this state after the power is turned on. If a system alarm is issued, the signal is set to “0”. The signal remains set to “1”, however, when an emergency stop or a similar operation is performed.

Servo Ready Signal SA<Fn000.6>

[Classification] Output signal

[Function] Signal SA turns to “1” when the servo system is ready to operate. On the machine side, for an axis that is to be braked, release the brake when this signal is “1” and apply the brake when this signal is “0”.
Time chart of this signal is as Fig. 2.2 (a).

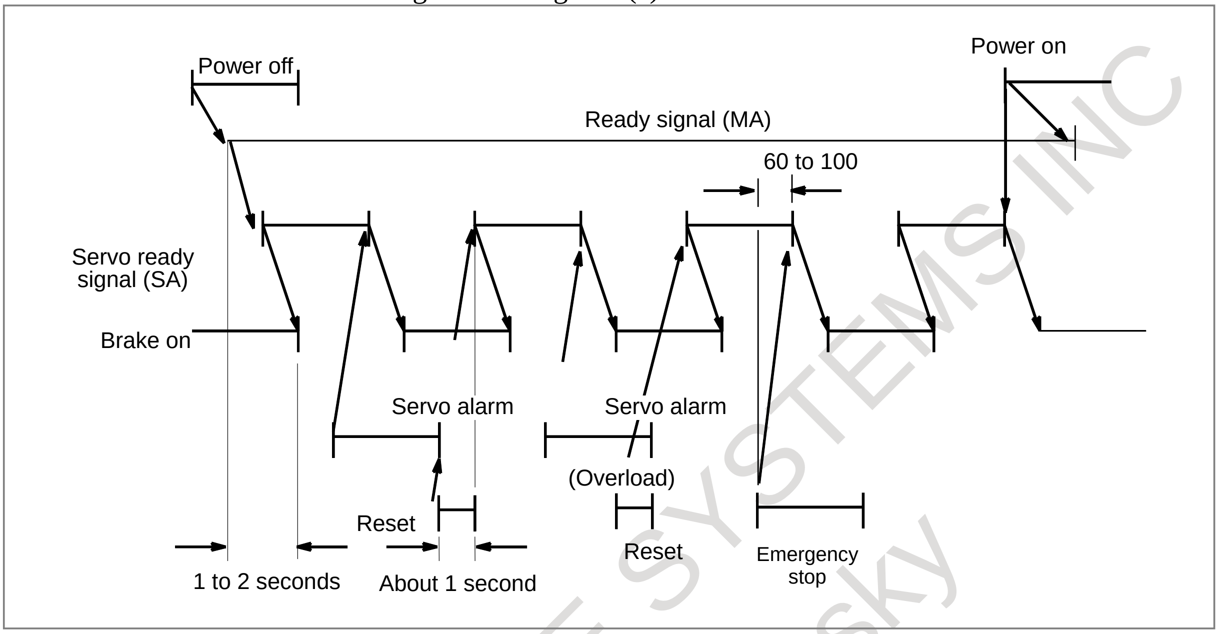


Fig. 2.2 (a) Time chart for servo ready signal

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn000		SA						
Fn001	MA							

2.3 OVERTRAVEL CHECK

2.3.1 Overtravel Signals

Overview

When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops as a result of tripping the limit switch, and an OVER TRAVEL is displayed.

Signal

Overtravel signals *+L1 to *+L8<Gn114>, *-L1 to *-L8<Gn116>

[Classification] Input signal

[Function] Indicates that the control axis has reached its stroke limit. There are individual signals for each direction in every control axis.
The +/- in the signal name indicates the direction and the number at the end indicates the control axis number.
+Lx

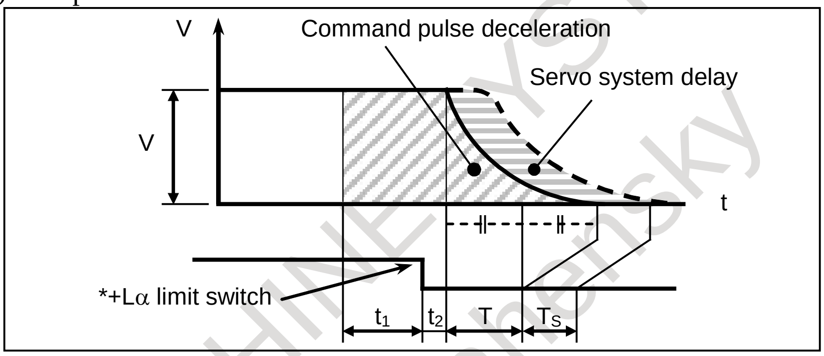
x : 1 : No. 1 axis is at stroke limit.
2 : No. 2 axis is at stroke limit.
3 : No. 3 axis is at stroke limit.
:
:
8 : No. 8 axis is at stroke limit.
+ : Limit reached in + direction.
- : Limit reached in - direction.

[Operation] When it is “0”, the control unit operates as given below.

- In automatic operation, if even one axis overtravel signal turns to “0”, all axes are decelerated to stop, an alarm is given and operation is halted.
- In manual operation, decelerates and stops movement of the axis "0" in the direction of "0" only, and the axis can be moved in the opposite direction.
- Once the axis overtravel signal has turned to “0”, the axis and direction is registered. Even if the signal returns to “1”, it is not possible to move that axis in that direction until the alarm is cleared.

The following shows the deceleration distance at overtravel.

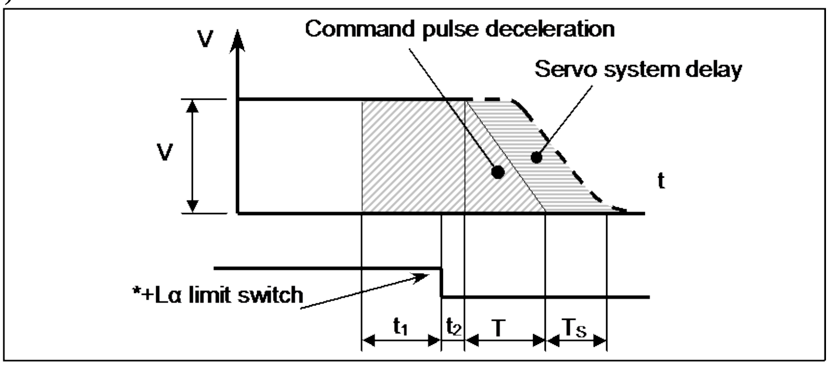
(i) Exponential acceleration/deceleration



$L=V(t_1+t_2+T+T_S)\times 1/60000$ [mm or inch]

L : Deceleration distance
V : Feedrate [mm/min or inch/min]
t_1 : Limit switch signal delay time (from limit switch operation to *+Lα signal turn off [msec])
t_2 : Receiver delay time 30msec
T : Acceleration/deceleration time [msec]
T_S : Servo system delay [msec]

(ii) Linear acceleration/deceleration



$L=V(t_1+t_2+T/2+T_S)\times 1/60000$ [mm or inch]

L : Deceleration distance
V : Feedrate [mm/min or inch/min]
t_1 : Limit switch signal delay time (from limit switch operation to *+Lα signal turn off [msec])
t_2 : Receiver delay time 30msec
T : Acceleration/deceleration time [msec]
T_S : Servo system delay [msec]

If rapid traverse is commanded and the bit 1 (LRP) of parameter No.1401 is 1, acceleration/deceleration time T is different depending on the traverse speed. T must set the value in which this is taken into consideration.

(iii) Bell-shaped acceleration/deceleration
It is similar to (ii).

> **NOTE**
> Servo system delay T_S is 33 msec when the servo unit is adjusted to the standard setting.

Releasing overtravel

First, move the tool into a safe zone under manual operation. Then press the reset button to reset the alarm.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn114	*+L8	*+L7	*+L6	*+L5	*+L4	*+L3	*+L2	*+L1
Gn116	*-L8	*-L7	*-L6	*-L5	*-L4	*-L3	*-L2	*-L1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3004			OTH					

[Input type] Parameter input
[Data type] Bit path

#5 OTH The overtravel signal is:
0: Checked
1: Not checked

> ⚠ **WARNING**
> For safety, usually set 0 to check the overtravel signal.

Alarm and message

Number	Message	Description
OT0506	+ OVERTRAVEL (HARD)	The stroke limit switch in the positive direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.
OT0507	- OVERTRAVEL (HARD)	The stroke limit switch in the negative direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Overtravel

2.3.2 Stored Stroke Check 1

Overview

A machine movable range is set with coordinates in the machine coordinate system in parameters. If the machine attempts to move beyond the range, it is decelerated and stopped and an alarm is displayed.
This function is enabled after manual reference position return is performed after power-on. (In case not using absolute pulse coder)
It can be used instead of an overtravel limit switch (hardware component).
When both functions are used at the same time, both are valid.
Unlike an overtravel limit switch, this function checks whether the position at which the machine is stopped after decelerated from the current position is beyond the range.
The stroke check 1 release signal common to all axes can be set to 1 so that the control unit does not make stroke check 1.

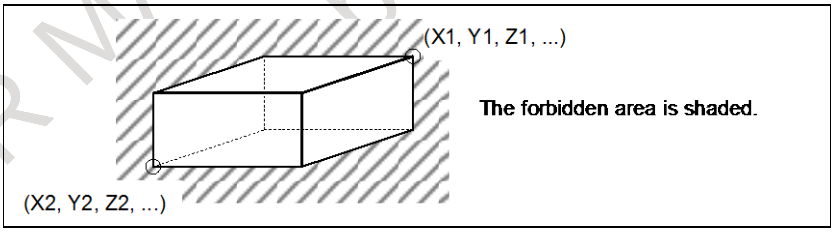


Fig. 2.3.2 (a)

Explanation

Parameters Nos. 1320, 1321 or Nos. 1326, 1327 set boundary. Outside the area of the set limits is a forbidden area. The machine tool builder usually sets this area as the maximum stroke.
When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.
At this time, a signal (overtravel alarm signals +OT1 to +OT8 <Fn124>, -OT1 to -OT8 <Fn126>) can be output to the PMC if bit 6 (OTS) of parameter No. 1301 is set to 1. In addition, when the tool enters the forbidden area during manual operation, the signal (overtravel alarm signal) can be output to the PMC without generating the alarm by setting bit 1 (NAL) of parameter No. 1300 to 1. With this parameter setting, the alarm is generated when the tool enters the forbidden area during automatic operation.
When bit 2 (LMS) of parameter No. 1300 is set to 1, stored stroke check 1 select signal EXLM <Gn007.6> selects stroke check 1-I (parameter No. 1320 and 1321) or stroke check 1-II (parameter No. 1326 and 1327).

And when bit 0 (DLM) of parameter No. 1301 is set to 1, axis direction dependent stored stroke check 1 switch signals +EXL1 to +EXL8<Gn104>, -EXL1 to -EXL8<Gn105> switches stroke limit 1-I (parameter No. 1320 and No. 1321) and stroke limit 1-II (parameter No. 1326 and No. 1327) for each axis direction.

Automatically released alarm

If bit 4 (OF1) of parameter No. 1301 is set to 1, when the axis moves within the movable area, the alarm can be canceled without reset operation.

NOTE

1 When a soft OT1 alarm is issued during automatic operation, the soft OT alarm may be issued each time operation restarts after the automatic release and the tool is moved along the axis to a position within the stroke limits through manual intervention, which can prevent the operation from being continued.
Example:
Suppose that a path N1→N2→N3 is programmed as shown in the Fig. 2.3.2 (b) and that an absolute command in the N2 block causes a soft OT alarm at point a. When automatic operation is restarted after the soft OT alarm is released automatically by retracting the tool to point b by manual intervention in the manual absolute mode, the tool moves toward point d, which is the end point of the N2 block, so the soft OT alarm is issued again at point c. It is impossible to proceed to the N3 block.

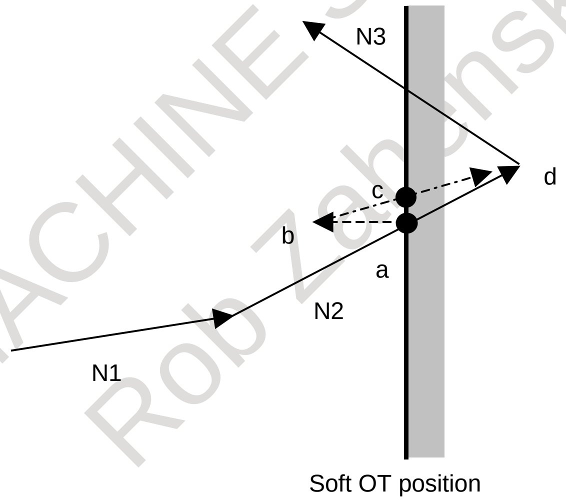


Fig. 2.3.2 (b)

2 The alarm can also be released automatically when the tool has entered the movable range by moving the tool along an axis by applying external force to the tool in the emergency stop state or servo-off state.

Simultaneous selection of stored stroke limit 1-I and -II

With the settings below, the inner area that is set for stored stroke check 1-II becomes a movement area, in addition to the inner area that is set for stored stroke check 1-I.

- Bit 2 of (LMS) of parameter No. 1300 = 1
- Bit 0 of (DLM) of parameter No. 1301 = 0
- Bit 1 of (LMA) of parameter No. 1301 = 1
- Stored stroke check 1 select signal EXLM <Gn007.6> = “1”

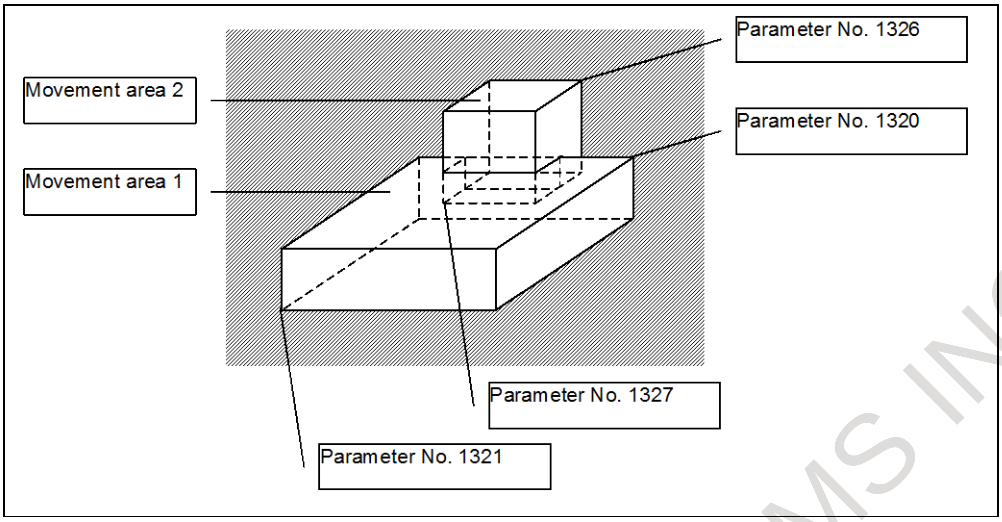


Fig. 2.3.2 (c)

> **NOTE**
> Simultaneous selection of stored stroke limit 1-I and -II cannot be used together with High precision oscillation function.

Signal

Stored stroke check 1 select signal EXLM<Gn007.6>

[Classification] Input signal

[Function] Selects stroke check 1-I (parameter Nos. 1320 and 1321) or stroke check 1-II (parameter Nos. 1326 and 1327).

[Operation] If this signal is “0”, the movement area for stroke check 1 is the inner area that is set for stroke check 1-I. (Area A in Fig. 2.3.2 (d))

If this signal is “1”, the movement area for stroke check 1 is as described below, depending on the setting of bit 1 (LMA) of parameter No. 1301.

- If bit 1 (LMA) of parameter No. 1301 is 0, the inner area that is set for stroke check 1-II. (Area B in Fig. 2.3.2 (d))
- If bit 1 (LMA) of parameter No. 1301 is 1, the inner area that is set for stored stroke check 1-II, in addition to the inner area set for stroke check 1-I. (Areas A and B in Fig. 2.3.2 (d))

This signal is effective if bit 2 (LMS) of parameter No. 1300 is 1.

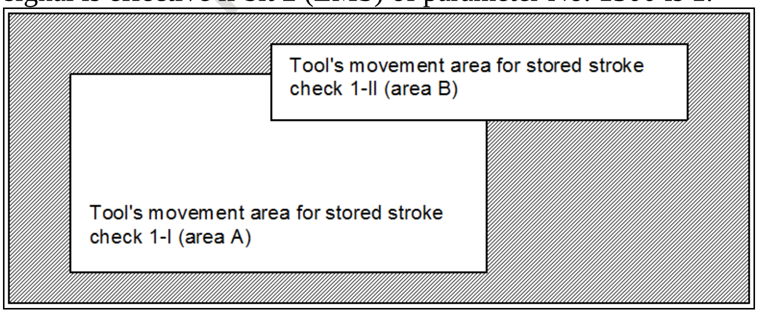


Fig. 2.3.2 (d)

Stroke check release signal RLSOT<Gn007.7>

[Classification] Input signal

[Function] Selects whether the stored stroke check 1 limits are checked or not.

[Operation] When this signal is set to “1”, does not check the stored stroke check 1 limits.

Axis direction dependent stored stroke check 1 switch signals +EXL1 to +EXL8<Gn104>, -EXL1 to -EXL8<Gn105>

[Classification] Input signal

[Function] Switches between stroke limit 1-I (parameter No. 1320 and No. 1321) and stroke limit 1-II (parameter No. 1326 and No. 1327) for each axis direction.
When this signal goes “1”, the CNC operates as described below.

1 +EXL1, +EXL2, . . .
Stroke check 1 (+ side) is checked using parameter No. 1326 instead of No. 1320.

2 -EXL1, -EXL2, . . .
Stroke limit 1 (- side) is checked using parameter No. 1327 instead of No. 1321.

Only if the bit 0 (DLM) of parameter No.1301 is set to 1, this signal is enabled.

Overtravel alarm signals +OT1 to +OT8<Fn124>, -OT1 to -OT8<Fn126>

[Classification] Output signal

[Function] Indicates whether the tool was about to enter the parameter-specified forbidden area (stored stroke limits).
This signal is valid when bit 1 (NAL) of parameter No. 1300 is set to 1 or when bit 6 (OTS) of parameter No. 1301 is set to 1. For differences in output condition between these parameter settings, see [Output cond.] below.
This signal is provided for each direction of each controlled axis. The sign + or - in the signal name indicates the direction, and the number at the end of the signal name indicates the number of a controlled axis.

* <u>x</u>OT<u>y</u>

y: 1 ….. No. 1 axis at stroke limit.
2 ….. No. 2 axis at stroke limit.
3 ….. No. 3 axis at stroke limit.

x: +: Positive direction
-: Negative direction

[Output cond.] If bit 1 (NAL) of parameter No. 1300 is set to 1, this signal is set to “1” when a move command in automatic operation or manual operation causes the tool to be about to enter the forbidden area (stored stroke limit 1) specified by parameter Nos. 1320 and 1321.
If bit 6 (OTS) of parameter No. 1301 is set to 1, this signal is set to “1” when one of the following overtravel alarms is issued:

- Alarms OT0500, OT0501 : Stored stroke check 1
- Alarms OT0502, OT0503 : Stored stroke check 2
- Alarms OT0504, OT0505 : Stored stroke check 3
- Alarms OT0506, OT0507 : Hard OT
- Alarms OT0510, OT0511 : Stored stroke check before movement

Once the signal is set to “1”, the axis and direction of the signal are stored. It is impossible to further move the tool along that axis in that direction. The tool can be moved along the axis only in the direction opposite to the stored direction. Moving the axis in the opposite direction resets the signal to “0”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn007	RLSOT	EXLM						
Gn104	+EXL8	+EXL7	+EXL6	+EXL5	+EXL4	+EXL3	+EXL2	+EXL1
Gn105	-EXL8	-EXL7	-EXL6	-EXL5	-EXL4	-EXL3	-EXL2	-EXL1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn124	+OT8	+OT7	+OT6	+OT5	+OT4	+OT3	+OT2	+OT1
Fn126	-OT8	-OT7	-OT6	-OT5	-OT4	-OT3	-OT2	-OT1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA					LMS	NAL	

[Input type] Setting input
[Data type] Bit path

#1 NAL When the tool enters the inhibition area of stored stroke limit 1:
0: The overtravel alarm signal is not output.
1: The overtravel alarm signal is output, and the tool is decelerated to a stop.
If manual operation is in progress at this time, the alarm is not output.

> **NOTE**
> Even if this parameter is set to 1, an alarm is displayed and the tool is decelerated to a stop when a move command issued during automatic operation causes the tool to get in a parameter-specified inhibition area. Also in this case, the overtravel alarm signal is output for the PMC.

#2 LMS The stored stroke check 1 select signal EXLM
0: Disabled
1: Enabled
When bit 0 (DLM) of parameter No. 1301 is set to 1, the stored stroke check 1 select signal EXLM <Gn007.6> is made invalid.

#7 BFA When the stored stroke check 1, 2, or 3 alarm is issued, an interference alarm is issued with the inter-path interference check function (T series), or a chuck/tail stock barrier (T series) alarm is issued:
0: The tool stops after entering the prohibited area.
1: The tool stops before the prohibited area.

> **NOTE**
> 1. This parameter is valid even in AI contour control.
> 2. This parameter is invalid for the slave axis under axis synchronous control.

	#7	#6	#5	#4	#3	#2	#1	#0
1301		OTS		OF1			LMA	DLM

[Input type] Setting input
[Data type] Bit path

#0 DLM The axis direction dependent stored stroke check 1 switch signals +EXLx and –EXLx are:
0: Disabled.
1: Enabled.
When this parameter is set to 1, the stored stroke check 1 select signal EXLM <Gn007.6> is made invalid.

#1 LMA When the stored stroke check 1 select signal EXLM <Gn007.6> is "1" with bit 2 (LMS) of parameter No. 1300 set to 1, the movable area for stored stroke check 1 is:
0: The inside area set for stored stroke check 1-II.
1: The inside area set for stored stroke check 1-I and the inside area set for stored stroke check 1-II as well.

#4 OF1 If the tool is moved into the range allowed on the axis after an alarm is raised by stored stroke check 1,
0: The alarm is not canceled before a reset is made.
1: The OT alarm is immediately canceled.

> **NOTE**
> In the cases below, the automatic release function is disabled. To release an alarm, a reset operation is required.
> 1 When a setting is made to issue an alarm before a stored stroke limit is exceeded (bit 7 (BFA) of parameter No. 1300 is set to 1)
> 2 When an another overtravel alarm (such as stored stroke check 2, stored stroke check 3, and interference check) is already issued
> 3 When an overtravel alarm is already issued with High precision oscillation function.

#6 OTS When the overtravel alarm is issued:
0: The overtravel alarm signal is not output to the PMC.
1: The overtravel alarm signal is output to the PMC.

1320	Coordinate value I of stored stroke check 1 in the positive direction on each axis
1321	Coordinate value I of stored stroke check 1 in the negative direction on each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate value of stored stroke check 1 on each axis in the + or - direction in the machine coordinate system.

> **NOTE**
> 1 Specify diameter values for any axes for which diameter programming is specified.
> 2 The area outside the area set by parameters Nos. 1320 and 1321 is a prohibited area.

1326	Coordinate value II of stored stroke check 1 in the positive direction on each axis

1327	Coordinate value II of stored stroke check 1 in the negative direction on each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the coordinate value of stored stroke check 1 on each axis in the + and - directions in the machine coordinate system.

When the stored stroke check switch signal EXLM is set to “1”, or the stored stroke check switch signals for each axis direction +EXLx and -EXLx are set to “1”, parameters Nos. 1326 and No. 1327 are used for stroke check instead of parameters Nos.1320 and No. 1321.

> **NOTE**
> 1 Specify diameter values for any axes for which diameter programming is specified.
> 2 The area outside the area set by parameter No. 1326 and No. 1327 is a prohibited area.
> 3 The EXLM signal is valid only when bit 2 (LMS) of parameter No. 1300 is set to 1.
> 4 When axis direction dependent stored stroke check 1 switch signals (+EXL1 to +EXL8, -EXL1 to -EXL8) is enabled (with bit 0 (DLM) of parameter No. 1301 set to 1), stroke parameter switching by the stored stroke check 1 select signal (EXLM3, EXLM2, EXLM) is disabled.

Alarm and message

Number	Message	Description
OT0500	+ OVERTRAVEL (SOFT 1)	A movement in the positive direction exceeded stored stroke check 1.
OT0501	- OVERTRAVEL (SOFT 1)	A movement in the negative direction exceeded stored stroke check 1.

Caution

> ⚠ **CAUTION**
> 1 If the two points for specifying a forbidden area are identical, all areas are handled as forbidden areas for stored stroke check 1.
> 2 The size of a forbidden area must be set correctly. If the size is set incorrectly, the stroke becomes infinite.

Note

NOTE

1. Bit 7 (BFA) of parameter No. 1300 can be used to specify whether to stop the tool before it enters the forbidden area or when it enters the area.
2. The stroke check is checked with finer accuracy than the displayed coordinate value. Therefore, even if the coordinates on the boundary are displayed, OT alarms may occur.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Stroke check

2.3.3 Stored Stroke Check 1 Area Expansion

Overview

In stored stroke check 1, up to eight different forbidden areas can be defined and selected.
Since the number of selectable forbidden areas increases, different forbidden areas can be used for different machine specifications.

Explanation

The boundary of the forbidden area in stored stroke check 1 is defined by specifying coordinate values I and II with parameters Nos. 1320, 1321, 1326, and 1327.

In this function, coordinate values III to VIII can be defined by parameters Nos. 1350 to 1361 in addition to coordinate values I and II above. The stored stroke check 1 select signal can be used to switch between these coordinate values of the forbidden area.

The relationship between the states of the signals and the stroke parameters (coordinate values) to be selected are shown below.

EXLM3	EXLM2	EXLM	Stroke parameter to be selected
0	0	0	Coordinate value I (No. 1320 / No. 1321)
0	0	1	Coordinate value II (No. 1326 / No.1327)
0	1	0	Coordinate value III (No. 1350 / No.1351)
0	1	1	Coordinate value IV (No. 1352 / No.1353)
1	0	0	Coordinate value V (No. 1354 / No.1355)
1	0	1	Coordinate value VI (No. 1356 / No.1357)
1	1	0	Coordinate value VII (No. 1358 / No.1359)
1	1	1	Coordinate value VIII (No. 1360 / No.1361)

Except for switching of forbidden areas by the signal selection above, the specification of stored stroke check 1 is used basically.
When the axis direction dependent stored stroke check 1 switch signals are enabled (when bit 0 (DLM) of parameter No.1301 is set to 1), this function is disabled.

Signal

Stored stroke check 1 select signals

EXLM<Gn007.6>, EXLM2<Gn531.6>, EXLM3<Gn531.7>

[Classification] Input signal
[Function] Switches between stroke parameters in stored stroke check 1 area expansion.
[Operation] The relationship between the states of stored stroke check 1 select signals EXLM3, EXLM2, and EXLM and the stroke parameters to be selected are shown below.

EXLM3	EXLM2	EXLM	Stroke parameter to be selected
0	0	0	Coordinate value I (No. 1320 / No. 1321)
0	0	1	Coordinate value II (No. 1326 / No.1327)
0	1	0	Coordinate value III (No. 1350 / No.1351)
0	1	1	Coordinate value IV (No. 1352 / No.1353)
1	0	0	Coordinate value V (No. 1354 / No.1355)
1	0	1	Coordinate value VI (No. 1356 / No.1357)
1	1	0	Coordinate value VII (No. 1358 / No.1359)
1	1	1	Coordinate value VIII (No. 1360 / No.1361)

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn007		EXLM						
Gn531	EXLM3	EXLM2						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300						LMS		

[Input type] Setting input
[Data type] Bit path

#2 LMS The stored stroke check 1 select signal (EXLM3, EXLM2, or EXLM when stored stroke check 1 area expansion is used) is
0: Disabled
1: Enabled

> **NOTE**
> When bit 0 (DLM) of parameter No. 1301 is set to 1, the stored stroke check 1 select signal EXLM (EXLM3, EXLM2, or EXLM when stored stroke check 1 area expansion is used) is made invalid.

1320	Coordinate value I of stored stroke check 1 in the positive direction on each axis
1321	Coordinate value I of stored stroke check 1 in the negative direction on each axis
1326	Coordinate value II of stored stroke check 1 in the positive direction on each axis
1327	Coordinate value II of stored stroke check 1 in the negative direction on each axis
1350	Coordinate value III of stored stroke check 1 in the positive direction on each axis
1351	Coordinate value III of stored stroke check 1 in the negative direction on each axis
1352	Coordinate value IV of stored stroke check 1 in the positive direction on each axis
1353	Coordinate value IV of stored stroke check 1 in the negative direction on each axis
1354	Coordinate value V of stored stroke check 1 in the positive direction on each axis
1355	Coordinate value V of stored stroke check 1 in the negative direction on each axis
1356	Coordinate value VI of stored stroke check 1 in the positive direction on each axis
1357	Coordinate value VI of stored stroke check 1 in the negative direction on each axis
1358	Coordinate value VII of stored stroke check 1 in the positive direction on each axis

1359	Coordinate value VII of stored stroke check 1 in the negative direction on each axis
1360	Coordinate value VIII of stored stroke check 1 in the positive direction on each axis
1361	Coordinate value VIII of stored stroke check 1 in the negative direction on each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the coordinate value of stored stroke check 1 on each axis in the + or - direction in the machine coordinate system.

The stored stroke check 1 select signal (EXLM3, EXLM2, EXLM) and the stroke parameter to be selected have the following relationships:

EXLM3	EXLM2	EXLM	Stroke parameter to be selected
0	0	0	Coordinate value I (No. 1320 / No. 1321)
0	0	1	Coordinate value II (No. 1326 / No. 1327)
0	1	0	Coordinate value III (No. 1350 / No. 1351)
0	1	1	Coordinate value IV (No. 1352 / No. 1353)
1	0	0	Coordinate value V (No. 1354 / No. 1355)
1	0	1	Coordinate value VI (No. 1356 / No. 1357)
1	1	0	Coordinate value VII (No. 1358 / No. 1359)
1	1	1	Coordinate value VIII (No. 1360 / No. 1361)

NOTE

1. Specify diameter values for any axes for which diameter programming is specified.
2. The outside of the area set with each parameter is treated as the inhibition area.
3. The stored stroke check 1 select signal (EXLM3, EXLM2, EXLM) is valid only when bit 2 (LMS) of parameter No. 1300 is 1.
4. When axis direction dependent stored stroke check 1 is enabled (with bit 0 (DLM) of parameter No. 1301 set to 1), stroke parameter switching by the stored stroke check 1 select signal (EXLM3, EXLM2, EXLM) is disabled.

Alarm and message

Number	Message	Description
OT0500	+ OVERTRAVEL (SOFT 1)	Exceeded the stored stroke check 1 when moving in the + direction.
OT0501	- OVERTRAVEL (SOFT 1)	Exceeded the stored stroke check 1 when moving in the - direction.

Note

NOTE

The basic notes are the same as those of stored stroke check 1.

2.3.4 Stored Stroke Check 2, 3

Overview

For stored stroke check 2, the outside or inside of the area specified by parameters or a program is defined as the forbidden area. As a limit position, specify a distance from the origin of the machine coordinate system. This function is enabled the after reference point establishes at power-on. When the limits are specified in a program, they can be set for the X-, Y-, and Z-axes. For this reason, the forbidden area can be changed according to the workpiece. Whether to define the inside or outside of the specified area as the forbidden area is determined by setting the corresponding parameter.

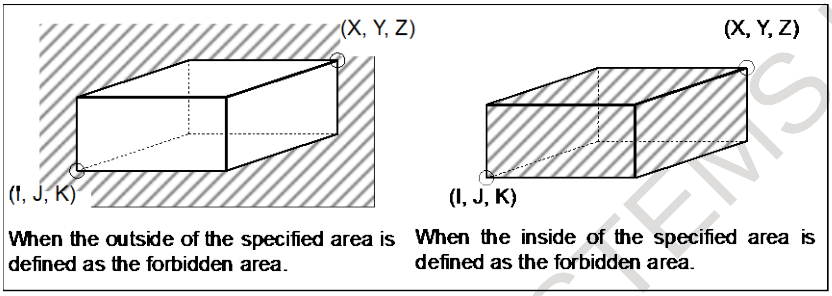


Fig. 2.3.4 (a) Stored stroke check 2

For stored stroke check 3, the inside of the area set by parameters is defined as the forbidden area.

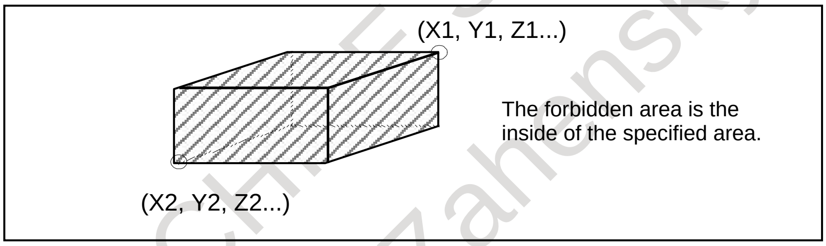


Fig. 2.3.4 (b) Stored stroke check 3

Explanation

Three areas which the tool cannot enter can be specified with stored stroke check 1, stored stroke check 2, and stored stroke check 3.

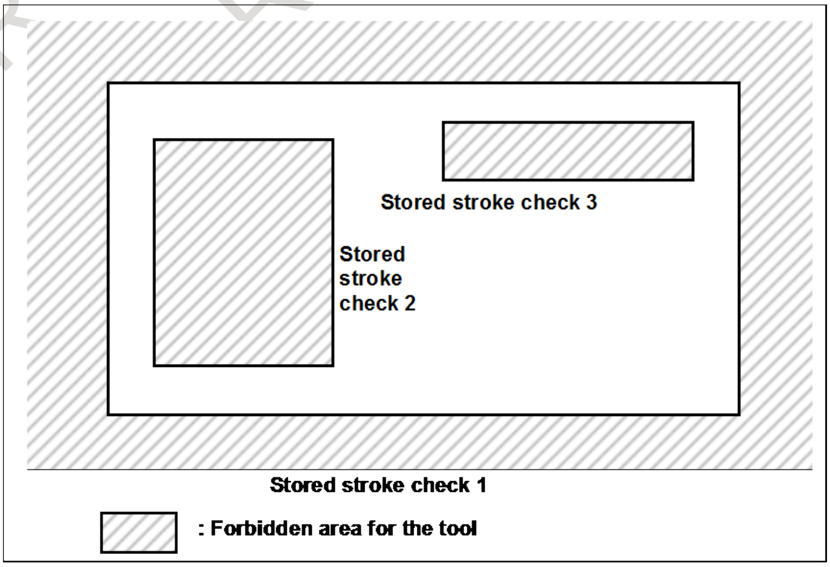


Fig. 2.3.4 (c) Stroke check

The following shows the areas which the tool cannot enter for each stored stroke check.

- Stroke check 1: Outside
- Stroke check 2: Outside or inside (switchable)
- Stroke check 3: Inside

When the tool moves into the forbidden area, an alarm is displayed and the tool is decelerated and stopped.
When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.

- Stored stroke check 2

Parameters Nos. 1322 and 1323 or commands set these boundaries. Inside or outside the area of the limit can be set as the forbidden area. Bit 0 (OUT) of parameter No. 1300 selects either inside or outside as the forbidden area.
In case of program command a G22 command forbids the tool to enter the forbidden area, and a G23 command permits the tool to enter the forbidden area.
Each of G22; and G23; should be commanded independently of another commands in a block.
The command below creates or changes the forbidden area:

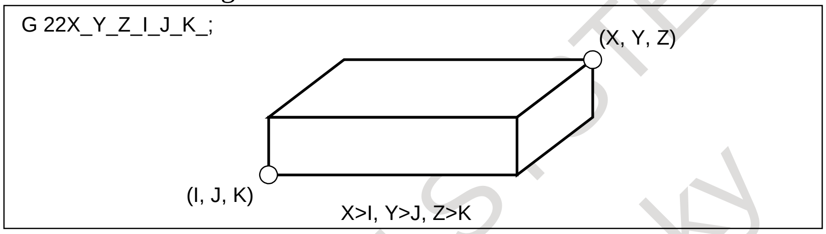


Fig. 2.3.4 (d) Creating or changing the forbidden area using a program

When setting the area by parameters, points A (Parameter No.1322) and B (Parameter No.1323) in the figure below must be set.

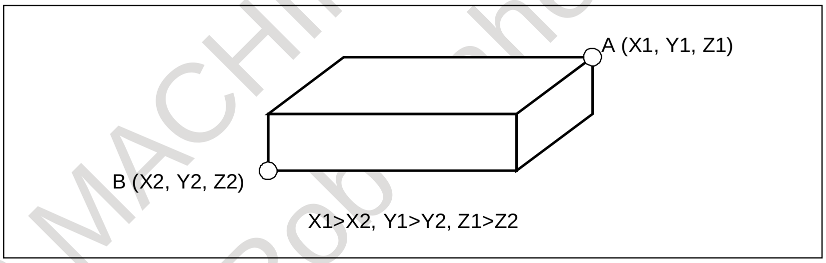


Fig. 2.3.4 (e) Creating or changing the forbidden area using a parameters

The values X1, Y1, Z1, X2, Y2, and Z2, which are set by parameters Nos. 1322 and 1323, must be specified by the distance from the machine coordinate system (machine increment). The values X, Y, Z, I, J, and K, which are set by a G22 command, must be specified by the distance in the least input increment (input increment).
When data is set by the program command, the command valued is converted in the machine increment and it set to the corresponding parameter (No.1322 and No.1323).

- Format

Stored stroke check 2 on

G22 X__ Y__ Z__ I__ J__ K__ ;

X, Y, Z:Coordinate of stored stroke check 2 in the positive direction on each axis
I, J, K: Coordinate of stored stroke check 2 in the negative direction on each axis

Use addresses X, Y, and Z or addresses I, J, and K. Addresses X and I specify the forbidden area related to the X-axis of the basic three axes. Addresses Y and J specify the forbidden area related to the Y-axis of the basic three axes. Addresses Z and K specify the forbidden area related to the Z-axis of the basic three axes. When addresses are omitted, a stroke check is made based on the values set by parameters.

Stored stroke check 2 off

G23 ;

- Stored stroke check 3

Set the boundary with parameters Nos. 1324 and 1325. The area inside the boundary becomes the forbidden area. The values X1, Y1, Z1, X2, Y2, and Z2 must be set as coordinates (machine increment) in the machine coordinate system.

- Checkpoint for the forbidden area

The parameter setting or programmed value (X, Y, Z, I, J, K) depends on which part of the tool or tool holder is checked for entering the forbidden area.
If point A is checked in Fig. 2.3.4 (f) and Fig. 2.3.4 (g), the machine coordinates value that corresponds to distance "a" should be set as the data for the stored stroke limit function. If point B is checked, the machine coordinates value that corresponds to distance "b" must be set. When checking the tool tip (like point A), and if the tool length varies for each tool, setting the forbidden area for the longest tool requires no re-setting and results in safe operation.

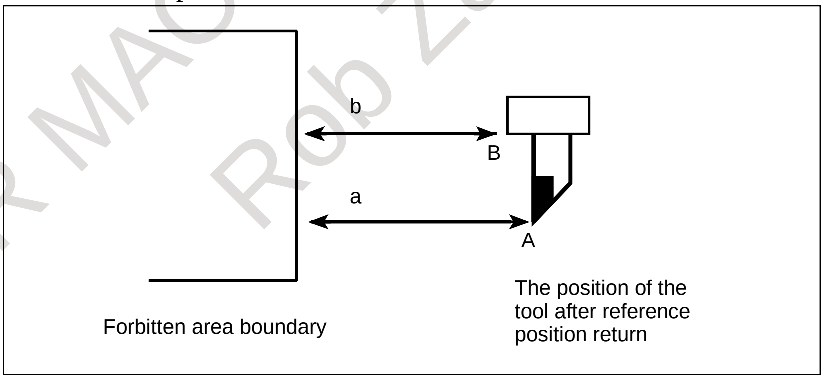


Fig. 2.3.4 (f) Setting the forbidden area (turning tool)

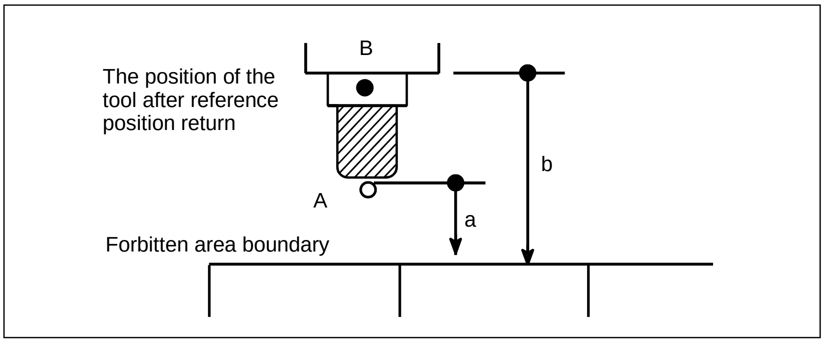
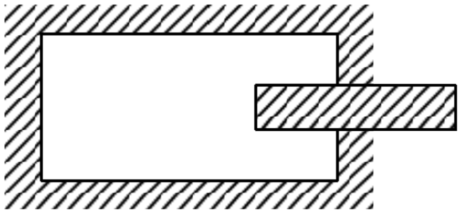


Fig. 2.3.4 (g) Setting the forbidden area (rotating tool)

- Forbidden area overlapping

Area can be set in piles.



Setting the forbidden area overlapping

Fig. 2.3.4 (h) Setting the forbidden area overlapping

Unnecessary limits should be set beyond the machine stroke.

- Condition under which each check is enabled

Each check becomes effective after the power is turned on and after reference point establishes.
After the power is turned on, if the reference position is in the forbidden area of each limit, an alarm is generated immediately. (Only in G22 mode for stored stroke check 2).

- Releasing the alarms

If the enters a forbidden area and an alarm is generated, the tool can be moved only in the backward direction. To cancel the alarm, move the tool backward until it is outside the forbidden area and reset the system. When the alarm is canceled, the tool can be moved both backward and forward.

- Change from G23 to G22 in a forbidden area

When G23 is switched to G22 in the forbidden area, the following results.
<1> When the forbidden area is inside, an alarm is informed in the next move.
<2> When the forbidden area is outside, an alarm is informed immediately.

- Timing for displaying an alarm

In stored stroke check 1,2,3, bit 7 (BFA) of parameter No. 1300 selects whether an alarm is displayed immediately before the tool enters the forbidden area or immediately after the tool has entered the forbidden area.

Signal

Stroke check 3 release signal RLSOT3<Gn007.4>

[Classification] Input signal
[Function] Selects whether stored stroke check 3 is checked.
[Operation] When this signal is set to “1”, does not check stored stroke check 3.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn007				RLSOT3				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA		RL3					OUT

[Input type] Setting input
[Data type] Bit path

#0 OUT The area inside or outside of the stored stroke check 2 is set as an inhibition area
0: Inside
1: Outside

#5 RL3 Stored stroke check 3 release signal RLSOT3 is
0: Disabled
1: Enabled

#7 BFA When the stored stroke check 1, 2, or 3 alarm is issued, an interference alarm is issued with the inter-path interference check function (T series), or a chuck/tail stock barrier (T series) alarm is issued:
0: The tool stops after entering the prohibited area.
1: The tool stops before the prohibited area.

NOTE
1. This parameter is valid even in AI contour control.
2. This parameter is invalid for the slave axis under axis synchronous control.

	#7	#6	#5	#4	#3	#2	#1	#0
1310							**OT3x**	**OT2x**

[Input type] Setting input
[Data type] Bit axis

#0 OT2x Stored stroke check 2 for each axis is :
0: Disabled
1: Enabled

#1 OT3x Stored stroke check 3 for each axis is :
0: Disabled
1: Enabled

1322	**Coordinate value of stored stroke check 2 in the positive direction on each axis**
1323	**Coordinate value of stored stroke check 2 in the negative direction on each axis**

[Input type] Setting input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate value of stored stroke check 2 on each axis in the + or - direction in the machine coordinate system.

NOTE
1 Specify diameter values for any axes for which diameter programming is specified.
2 Whether the inside area or outside area is a prohibited area is set using bit 0 (OUT) of parameter No. 1300.

1324	Coordinate value of stored stroke check 3 in the positive direction on each axis

1325	Coordinate value of stored stroke check 3 in the negative direction on each axis

[Input type] Setting input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the coordinate value of stored stroke check 3 on each axis in the + or - direction in the machine coordinate system.

NOTE
1 Specify diameter values for any axes for which diameter programming is specified.
2 The area inside the area set by parameters Nos. 1324 and 1325 is a prohibited area.

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23							

[Input type] Parameter input
[Data type] Bit path

#7 G23 When the power is turned on
0: G22 mode (stored stroke check 2 on)
1: G23 mode (stored stroke check 2 off)

	#7	#6	#5	#4	#3	#2	#1	#0
10345							L2D	

[Input type] Parameter input
[Data type] Bit

#1 L2D When the forbidden area of the stored stroke check 2, 3 (Parameter No.1322, 1323) is set or it is changed by G22 command, the setting value for axes with diameter specification is:
0: Half of command value
1: Command value

Alarm and message

Number	Message	Description
OT0502	+ OVERTRAVEL (SOFT 2)	A movement in the positive direction exceeded stored stroke check 2. Alternatively, in the chuck and tail stock barrier, an entry to the forbidden area was made during movement in the positive direction.
OT0503	- OVERTRAVEL (SOFT 2)	A movement in the negative direction exceeded stored stroke check 2. Alternatively, in the chuck and tail stock barrier, an entry to the forbidden area was made during movement in the negative direction.
OT0504	+ OVERTRAVEL (SOFT 3)	A movement in the positive direction exceeded stored stroke check 3.
OT0505	- OVERTRAVEL (SOFT 3)	A movement in the negative direction exceeded stored stroke check 3.

Caution

⚠ **CAUTION**

1 If the two points for specifying a forbidden area are identical, all areas are handled as movable areas in case of stored stroke check 2 that the inside area is forbidden (bit 0 (OUT) of parameter No. 1300 = 0), or stored stroke check 3. And all areas except the specified point are handled as forbidden areas in case of stored stroke check 2 that the outside area is forbidden (the parameter OUT = 1).

2 Even if the two points for specifying a forbidden area are erroneously set, the rectangular parallelepiped having the points as vertices is assumed as a boundary.

3 Since an axis without the reference position return function has no forbidden areas, there are no alarms about forbidden areas for the axis.

Note

NOTE

1 For checks 1,2 and 3, bit 7 (BFA) of parameter No. 1300 can be used to specify whether to stop the tool before it enters the forbidden area or when it enters the area.

2 Because checking is made with higher precision than the precision of indicated coordinates, an OT alarm may be issued even when coordinates on a boundary is indicated.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Stroke check

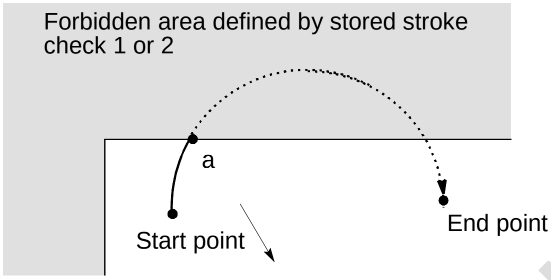
2.3.5 Stroke Limit Check before Move

During automatic operation, before the movement specified by a given block is started, whether the tool enters the forbidden area defined by stored stroke check 1, 2, or 3 is checked by determining the position of the end point from the current position of the machine and a specified amount of travel. If the tool is found to enter the forbidden area defined by a stored stroke limit, the tool is stopped immediately upon the start of movement for that block, and an alarm is displayed.

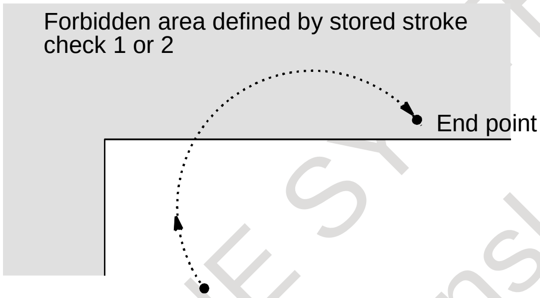
⚠ CAUTION

Whether the coordinates of the end point, reached as a result of traversing the distance specified in each block, are in a forbidden area is checked. In this case, the path followed by a move command is not checked. However, if the tool enters the forbidden area defined by stored stroke check 1, 2, or 3, an alarm is issued.

Example 1)

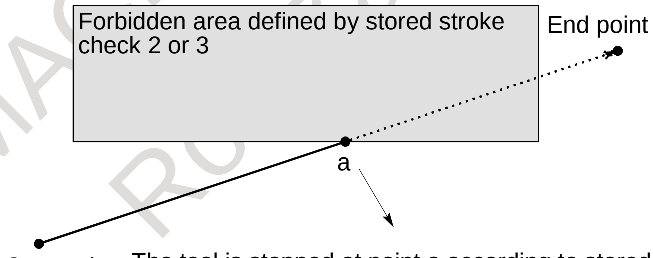


The tool is stopped at point a according to stored stroke check 1 or 2.

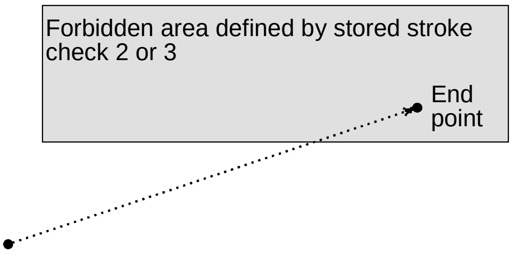


Immediately upon movement commencing from the start point, the tool is stopped to enable a stroke limit check before moving to be performed before movement.

Example 2)



The tool is stopped at point a according to stored stroke check 2 or 3.



Immediately upon movement commencing from the start point, the tool is stopped to enable a stroke limit check before moving to be performed before movement.

Explanation

This function is enabled by setting bit 7 (PLC) of parameter No. 1301 to 1.
When a stroke limit check before moving is performed, whether to check the movement performed by a G31 (skip) block and G37 (automatic tool length measurement) block can be determined using (bit 2 (NPC) of parameter No. 1301).

Limitation

- Machine lock

If machine lock is applied at the start of movement, no stroke limit check made before movement is performed.

- G23

When stored stroke check 2 is disabled (G23 mode), no check is made to determine whether the tool enters the forbidden area defined by stored stroke check 2.

- Program restart

When a program is restarted, an alarm is issued if the restart position is within a forbidden area.

- A block consisting of multiple operations

If a block consisting of multiple operations (such as a canned cycle) is executed, an alarm is issued at the start point of any operation whose end point falls within an inhibited area.

- Cylindrical interpolation mode

In cylindrical interpolation mode, no check is made.

T

- Polar coordinate interpolation mode

In polar coordinate interpolation mode, no check is made.

- 3-dimensional coordinate system conversion

In 3-dimensional coordinate system conversion mode, no check is made.

- PMC axis control

No check is made for a movement based on PMC axis control.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1301	PLC					NPC		

[Input type] Setting input
[Data type] Bit path

#2 NPC As part of the stroke limit check performed before movement, the movement specified in G31 (skip) and G37 (automatic tool length measurement) blocks is:
0: Checked
1: Not checked

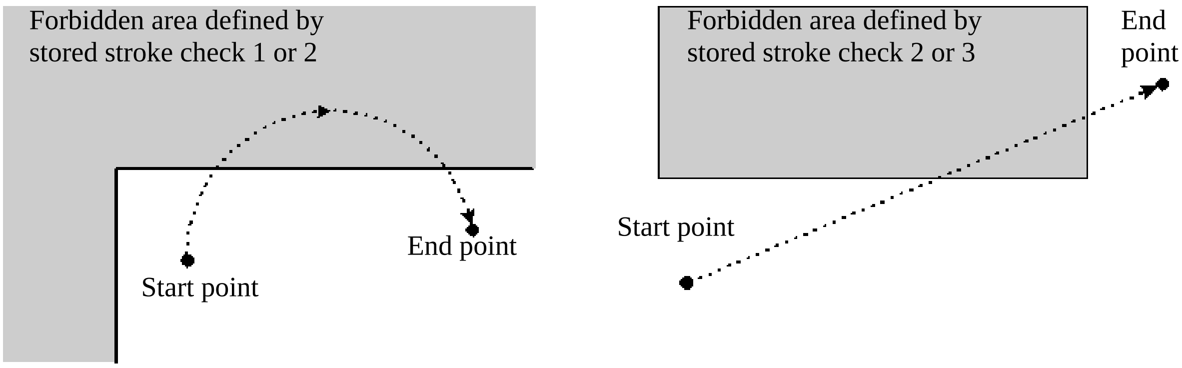
#7 PLC Stroke check before movement is:
0: Not performed
1: Performed

Alarm

Number	Message	Description
OT0510	+ OVERTRAVEL (PRE-CHECK)	The block end point was found in the + side stroke limit prohibition area during a stroke check before movement. Modify the program.
OT0511	- OVERTRAVEL (PRE-CHECK)	The block end point was found in the - side stroke limit prohibition area during a stroke check before movement. Modify the program.

2.3.6 Check of the Tool Path between Blocks by Stroke Limit Check before Move

By setting bit 0 (SBA) of parameter No.1302 to 1, whether the tool enters the forbidden area defined by Stored stroke limit 1, 2, or 3 is checked on the tool path of movement command in addition to checking the end point. (Hereinafter, this check is called “check of the tool path between blocks”.)
Check of the tool path between blocks is effective in Positioning (G00), Liner interpolation (G01), Circular interpolation (G02, G03), High-speed reference position return (G28, G30), or Selecting a machine coordinate system (G53).
If the tool enters the forbidden area defined by stored stroke check 1, 2, or 3, an alarm is issued when check of the tool path between blocks is not effective.



When forbidden area is outside. **When forbidden area is inside.**

Immediately upon movement commencing from the start point, the tool is stopped to enable a stroke limit check before moving to be performed before movement.

: Forbidden area

Fig. 2.3.6 (a) Check of the tool path between blocks

Explanation

Plane selection axes

When one axis or both axes among the selection axes of selected plane (G17, G18 or G19 plane) not exist (Hereinafter, this selection axes are called “plane selection axes”.), check of the tool path between blocks is not executed.

Target axis of check of the tool path between blocks

Check of the tool path between blocks is executed on the 3-dimension which is composed by the two axes of plane selection axes and the one axis other than plane selection axes (Hereinafter, this axis is called “plane non-selection axis”).

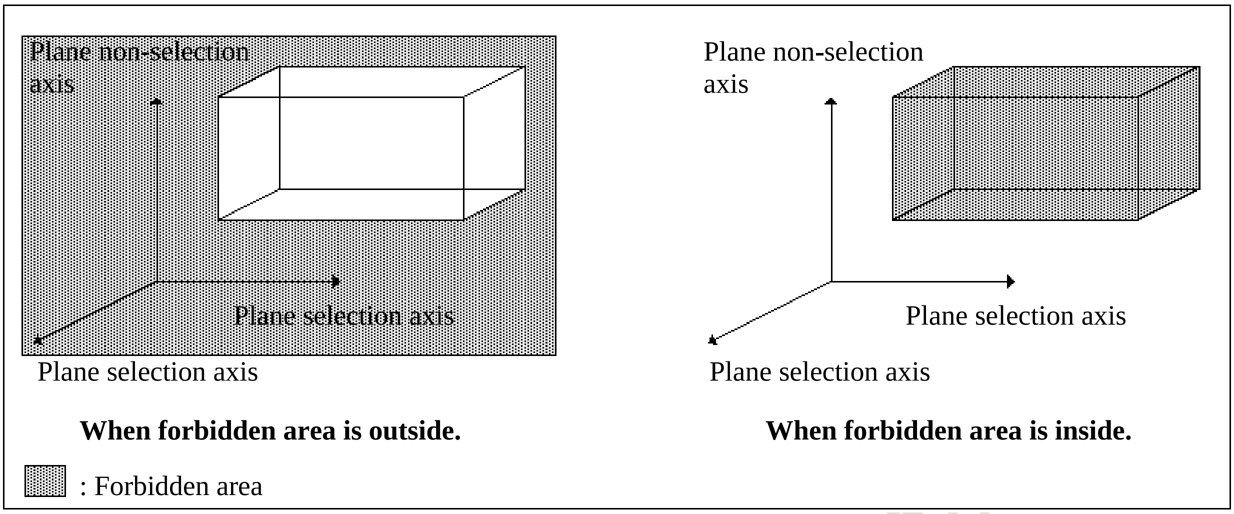


Fig. 2.3.6 (b) Target axis of check of the tool path between blocks

Normally, the basic axis other than plane selection axes in basic coordinate system is set to plane non-selection axis.

Example 1)
When X-Y plane (G17) is selected with the following axis configuration, plane selection axes are X axis and Y axis, and plane non-selection axis is Z axis. (Fig. 2.3.6 (c))
No.1022
(X)=1: X axis of the basic three axes
(Y)=2: Y axis of the basic three axes
(Z)=3: Z axis of the basic three axes
(W)=7: Axis parallel to Z axis
(A)=0: Rotary axis

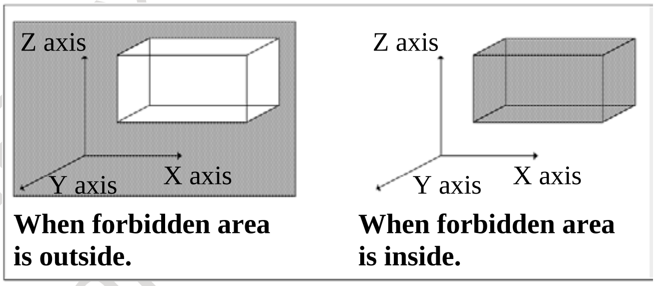


Fig. 2.3.6 (c) Target axis in Example 1

However, there are the following exceptions.

(1) The basic axis other than plane selection axes in basic coordinate system does not exist.

(1) - 1 When parallel axis does not exist.
In this case, check of the tool path between blocks is executed on the plane composed by plane selection axes.

Example 2)
When X-Y plane (G17) is selected with the following axis configuration, plane selection axes are X axis and Y axis, and plane non-selection axis does not exist. (Fig. 2.3.6 (d))
No.1022
(X)=1: X axis of the basic three axes
(Y)=2: Y axis of the basic three axes
(A)=0: Rotary axis

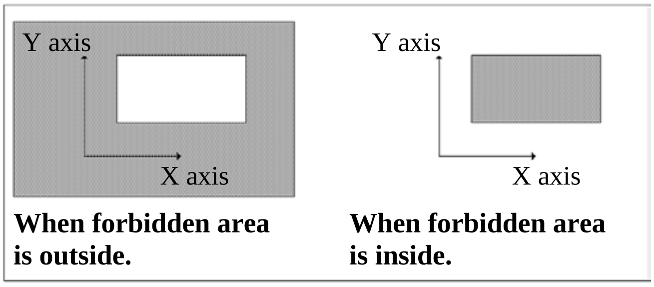


Fig. 2.3.6 (d) Target axis in Example 2

(1) - 2 When the parallel axis exists.

In this case, the parallel axis of lowest number of intra-path relative axis number which is parallel to the basic axis other than plane selection axes is set to plane non-selection axis.

Example 3)

When X-Y plane (G17) is selected with the following axis configuration, plane selection axes are X axis and Y axis, and plane non-selection axis is W axis. (Fig. 2.3.6 (e))

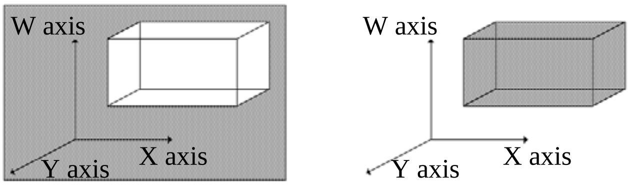
No.1022

(X)=1: X axis of the basic three axes

(Y)=2: Y axis of the basic three axes

(W)=7: Axis parallel to Z axis

(A)=0: Rotary axis



When forbidden area is outside. **When forbidden area is inside.**

Fig. 2.3.6 (e) Target axis in Example 3

(2) In the state of selecting the plane which includes the plane non-selection axis direction, the axis movement had been specified.

In this case, the axis which was specified by the plane selection is set to plane non-selection axis. However, when the axis movement by "List of modes that cannot execute check of the tool path between blocks" or "List of commands that cannot execute check of the tool path between blocks" is specified, the axis which was specified by the plane selection is not set to plane non-selection axis. And, the basic axis other than plane selection axes in the basic coordination system is set to plane non-selection axis when G code group of plane selection G code clear by reset etc. (Example 4)

Example 4)

When U-Y plane (G17) is selected after specifying the axis movement in W-X plane (G18) with the following axis configuration (Program example 1), plane selection axes are U axis and Y axis, and plane non-selection axis is W axis. (Fig. 2.3.6 (f))

No.1022

(X)=1: X axis of the basic three axes

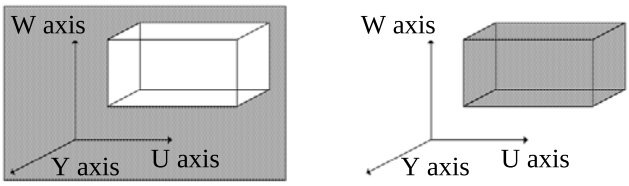
(Y)=2: Y axis of the basic three axes

(Z)=3: Z axis of the basic three axes

(U)=5: Axis parallel to X axis

(W)=7: Axis parallel to Z axis

(A)=0: Rotary axis



When forbidden area is outside. **When forbidden area is inside.**

Fig. 2.3.6 (f) Target axis in Example 4 (U-Y-W)

However, plane non-selection axis is Z axis in the following case. (Fig. 2.3.6 (g))

1. After selecting W-X plane (G18), U-Y plane (G17) is selected without specifying the axis movement (Program example 2).
2. After selecting W-X plane (G18), U-Y plane (G17) is selected after specifying only the axis movement by "List of modes that cannot execute check of the tool path between blocks" or "List of commands that cannot execute check of the tool path between blocks" (Program example 3).

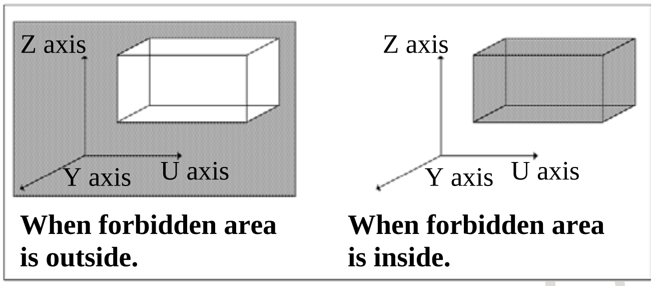


Fig. 2.3.6 (g) Target axis in Example 4 (U-Y-Z)

When G code group of plane selection G code clears by reset etc..

Afterwards, when G code group of plane selection G code clears by reset etc., plane selection axes are X axis and Y axis, and plane non-selection axis is Z axis. (Fig. 2.3.6 (h))

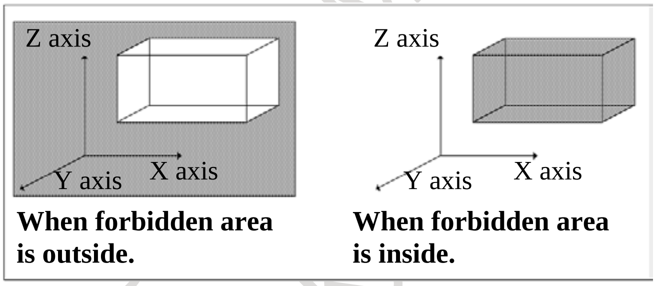


Fig. 2.3.6 (h) Target axis in Example 4 (Clear state)

<Program example 1>
O0001
G18W0X0
G01W10.0X10.0F1000.0 } When axis movement of W axis is specified,
:
G17U0Y0
:
: } plane selection axes are U axis and Y axis, and plane non-selection axis is W axis.

<Program example 2>
O0001
G18W0X0
:
: } When axis movement of W axis is not specified,
G17U0Y0
:
: } plane selection axes are U axis and Y axis, and plane non-selection axis is Z axis.

<Program example 3>
O0001
G18W0X0
G31W100.0F100.0
G31X100.0F100.0 } When axis movement of W axis is specified by G31 command in case of setting bit 2 (NPC) of parameter No.1301 to 1 or etc. ,
G17U0Y0
:
: } plane selection axes are U axis and Y axis, and plane non-selection axis is Z axis.

NOTE

1. Check of the tool path between blocks is executed for forbidden area defined by stored stroke limit of plane selection axes and plane non-selection axis. Therefore, even if the axis other than plane selection axes and plane non-selection axis exceed the forbidden area defined by stored stroke limit in the tool path between blocks, the alarm is not generated by check of the tool path between blocks. (However, because stored stroke check of normal is effective, the alarm is generated when approaching the forbidden area defined by stored stroke limit during moving the axis.)
2. When the forbidden area is defined by inside of stored stroke limit, the alarm is generated if all axes to which stored stroke check is effective enter the forbidden area. However, in check of the tool path between blocks, the alarm is generated if plane selection axes and plane non-selection axis enters the forbidden area.

Limitation

Limitation of Stroke limit check before move

Limitation of Stroke limit check before move is effective for check of the tool path between blocks.

Non-linear type positioning

Check of the tool path between blocks is executed for the straight-line that connect the start position and end position of the block even if Positioning (G00), High-speed reference position return (G28, G30), or Selecting a machine coordinate system (G53) is move by non-linear type. Therefore, check of the tool path between blocks might not be able to be executed correctly, depending on the command.

Circular interpolation that is imperfect circle

When Circular interpolation that is imperfect circle is specified, check of the tool path between blocks is executed for the base circle. Therefore, check of the tool path between blocks might not be able to be executed correctly, depending on the shape of Circular interpolation.

Machine lock

For the axis which is Machine lock state when starting the block, check of the tool path between blocks is not executed.

Stored stroke check 2 off (G23)

When Stored stroke check 2 is invalid (G23 mode), check of the tool path between blocks for the forbidden area defined by Stored stroke limit 2 is not executed.

Arbitrary angular axis control

Stored stroke limit can be checked in Cartesian coordinate system by setting bit 0 (AOT), bit 1 (AO2) or bit 2 (AO3) of parameter No.8201 in Arbitrary angular axis control. By setting of checking stored stroke limit in Cartesian coordinate system, check of the tool path between blocks is effective. Check of the tool path between blocks by this stored stroke limit is not executed if angular axis or perpendicular axis to which stored stroke limit is checked by slanted coordinate system is set to plane selection axes or plane non-selection axis.

List of modes that cannot execute check of the tool path between blocks

Check of the tool path between blocks is not executed during the mode of following functions.

T

- Polar coordinate interpolation

- Cylindrical interpolation
- 3-dimensional coordinate conversion
- Skip function (However, the case of setting bit 2 (NPC) of parameter No.1301 to 0 is excluded.)
- Automatic tool length measurement (M series) / automatic tool offset (T series) (However, the case of setting bit 2 (NPC) of parameter No.1301 to 0 is excluded.)

List of commands that cannot execute check of the tool path between blocks

Check of the tool path between blocks is not executed by the following commands.

- Movement command by Real time macro commands
- Movement command by PMC axis control
- Movement command for slave axis of Tandem control

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1302								SBA

[Input type] Parameter input
[Data type] Bit path

#0 SBA In Stroke limit check before move, check of the tool path between blocks for movement command is, :

0: Not executed.
1: Executed.

> **NOTE**
> 1 Check of the tool path between blocks is invalid for the functions to which Stroke limit check before move is invalid.
> 2 Limitation of Stroke limit check before move is valid for check of the tool path between blocks.

Alarm and message

Number	Message	Description
OT0510	+ OVERTRAVEL (PRE-CHECK)	The block end point or the tool path between blocks was found in the + side stroke limit forbidden area during Stroke limit check before move. Modify the program.
OT0511	- OVERTRAVEL (PRE-CHECK)	The block end point or the tool path between blocks was found in the - side stroke limit forbidden area during Stroke limit check before move. Modify the program.

2.3.7 Checking the Stored Stroke during the Time from Power–on to the Reference Position Establishment

This function stores the machine coordinates present immediately before the power is turned off. Therefore, immediately after the power is turned on again, this function can restore the approximate machine coordinates and enables the function for checking the stored stroke during the time from power–on to the reference position establishment. (If bit 0 (DOTx) of parameter No. 1311 is set to 1)

Even before the reference position is established by manual reference position return, the stored stroke check can be performed using approximate machine coordinates.

Absolute coordinates are set based on the machine coordinates.
Data such as workpiece offsets specified by G92, G52, and so on before the power is turned off, however, is not set.
If bit 3 (PPD) of parameter No. 3104 is set to 1, the relative position indication is preset.

The stored stroke check is enabled immediately after the power is turned on.
This stored stroke check is performed in the same way as the normal stored stroke check.

The reference position is not established until a reference position return takes place.
(Until then, reference position establishment signals ZRF1 to ZRF8<Fn120> are “0”.
Precision compensation functions such as pitch error compensation are not performed.)

- Axis synchronous control

In case of the synchronization establishment of the axis synchronous control, when the position is set by this function, the synchronization establishment is executed.

NOTE

1. This function stores machine coordinates periodically while the CNC is operating. This means that no machine coordinates are stored when the tool moves while the power is off.
   If the power to the CNC is turned off while the tool is moving along an axis, correct machine coordinates cannot be stored.
   To reflect movements made while the power is off, use temporary absolute coordinate setting.
2. Because this function stores machine coordinates by using software, load is imposed on the system.
   For axes for which this function is not required, this function should not be set.
3. When an absolute position detector (absolute Pulsecoder) is used for an axis, this function cannot be used for that axis.
4. If the tool is already beyond a stored stroke immediately after the power is turned on, an overtravel alarm (OT alarm) is issued.
   In this case, release the alarm by moving the tool along the axis in the direction that does not cause overtravel.
   If bit 6 (LZR) of parameter No. 1300 is set to 1, the function for checking the stored stroke during the time from power–on to the reference position establishment is disabled.
5. When the power is turned on while the  key and the <P> address key are held down, the stored stroke check immediately after power-on is disabled.
   (When the MDI keys are provided as standard keys)
   (When performing this operation, exercise special care.)
   In this case, machine coordinates and absolute coordinates are set.
   When the compact type MDI keys are used, the stored stroke immediately after power-on is disabled by holding down the  key and <O> address key at power-on.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300		LZR						

[Input type] Setting input
[Data type] Bit path

#6 LZR When the stored stroke check immediately after power-on is enabled (bit 0 (DOTx) of parameter No. 1311 is set to 1), the stored stroke check is:
0: Performed even before a manual reference position return is made.
1: Not performed until a manual reference position return is made.

	#7	#6	#5	#4	#3	#2	#1	#0
1311								DOTx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 DOTx Stored stroke limit check immediately after power-on is:
0: Disabled.
1: Enabled.

If the stored stroke check is enabled, the machine coordinate value present immediately before the power is turned off is stored.
The machine coordinate value is set immediately after the power is turned on.
Based on the machine coordinate value, absolute coordinate and relative coordinate values are set.

> **NOTE**
> Because this function uses software to store machine coordinates, the function puts an extra load on the system. So, this function should not be set for axes that do not require this function. The amount of a movement made while the power is off is not reflected in machine coordinates immediately after the power is turned on.

2.3.8 Stroke Limit External Setting

Overview

When a tool is changed, the tool tip is aligned with the end of the limit area and signals are input. This operation sets the machine position (machine coordinates) at that time as the limit position in the parameters for coordinate values I of stored stroke check 1 (Nos.1320 and 1321). A setting signal is provided for each direction of each axis.

Signal

Stroke limit external setting signal +LM1 to +LM8<Gn110>, -LM1 to -LM8<Gn112>

[Classification] Input signal
[Function] Changes the values of the parameters for coordinate values I of stored stroke check 1 (Nos. 1320 and 1321).

[Operation] When these signals are set to "1", the control unit changes the values of the parameters Nos. 1320 and 1321 by using the machine coordinates when the signals are input as the stroke check values.

> **NOTE**
> Stroke limit external setting signal +LM1 to +LM8<Gn110>, -LM1 to -LM8<Gn112> should be set to "1" when the relevant axis is stopped. If these signals are set to "1" during axes movement, stroke limit may not be precisely.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn110	+LM8	+LM7	+LM6	+LM5	+LM4	+LM3	+LM2	+LM1
Gn112	-LM8	-LM7	-LM6	-LM5	-LM4	-LM3	-LM2	-LM1

Note

> **NOTE**
> 1 This function can not change the values of the parameters for coordinate values II to VIII of stored stroke check 1 (Nos.1326, 1327 and 1350 to 1361).

2.3.9 Stroke Limit Area Changing Function

Overview

This function can be used to rewrite the parameters that set the + side coordinate value and the - side coordinate values of the stroke limits even when the axis is traveling. The parameters can be rewritten by the PMC window function (WINDW: SUB52), FOCAS2, and a C Language Executor. The new stroke limit range is enabled immediately after the parameters are rewritten by any of these functions. The rewritable parameters are shown below.

- Parameter No. 1320:Coordinate value I of stored stroke check 1 in the positive direction on each axis
- Parameter No. 1321:Coordinate value I of stored stroke check 1 in the negative direction on each axis
- Parameter No. 1322:Coordinate value of stored stroke check 2 in the positive direction on each axis
- Parameter No. 1323:Coordinate value of stored stroke check 2 in the negative direction on each axis
- Parameter No. 1324:Coordinate value of stored stroke check 3 in the positive direction on each axis
- Parameter No. 1325:Coordinate value of stored stroke check 3 in the negative direction on each axis
- Parameter No. 1326:Coordinate value II of stored stroke check 1 in the positive direction on each axis
- Parameter No. 1327:Coordinate value II of stored stroke check 1 in the negative direction on each axis

The machining cycle time can be reduced because this function can rewrite parameters even if some axes are moving.

This function is enabled by setting bit 0 (SLM) of parameter No.1312 to 1.

Limitation

- Axis synchronous control

This function is disabled when the automatic slave axis parameter setting is enabled (bit 4 (SYP) of parameter No. 8303 = 1) in feed axis synchronous control. (Writing to the parameters is allowed only when all axes are stopped.)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1312								SLM

[Input type] Parameter input
[Data type] Bit

#0 SLM The stroke limit area changing function is:
0: Disabled.
1: Enabled.

Note

NOTE
If the parameters are rewritten so that the current position is included in a forbidden area during axis movement, the axis decelerates and stops, and an alarm is displayed. If an alarm occurs when the tool enters a forbidden area, the tool can move only opposite to the direction from which the tool came. After moving it outside the forbidden area in the opposite direction, release the alarm with a reset. When the alarm is released, the tool can move in both directions.

2.3.10 Stored stroke limit range switching function by signal

Overview

The range stored stroke limit can be switched by input signal of PMC. Therefore, the range stored stroke limit can easily be set again.

Explanation

- Stored stroke limit range switching

Stored stroke limit range switching function consists of the next five steps.

1) Data for switching is set.
Data for switching is set to data table (D) of PMC.
Data for switching is set from the address specified by parameter No.1313 and stored stroke limit range switching data selection signals OTD0 to OTD15<Gn594, Gn595>.

2) The offset from the head of data table (D) of PMC to which data for switching is set is specified.
The offset from the head of data table (D) of PMC that sets data for switching is specified by stored stroke limit range switching data selection signals.

3) The switched axis is specified.
The axis switched by stored stroke limit range switching axis selection signals OTA1 to OTA8 <Gn596> is specified. Two or more axes can be specified. Be careful of the compatibility of specification of axes and data for switching.

4) The switched kind of stored stroke limit is specified.
The kind of stored stroke limit is specified with stored stroke limit range switching selection signals +/-OT11,+/-OT12,+/-OT2,+/-OT3 <Gn597>. Two or more kinds can be specified at the same time.

5) Request and finish for switching
When above-mentioned steps from 1) to 4) have been prepared, stored stroke limit range switching start signal OTSW <Gn599.0> is set to "1". When stored stroke limit range switching finish signal OTSWFN <Fn599.0> becomes "1", the range of stored stroke limit of the specified axis is switched, therefore set stored stroke limit range switching start signal to "0". Other input signals can be specified until turning on stored stroke limit range switching start signal next time. The state of the signal can be held when you need not change the specified signal.

- Stored stroke limit range switching cancellation

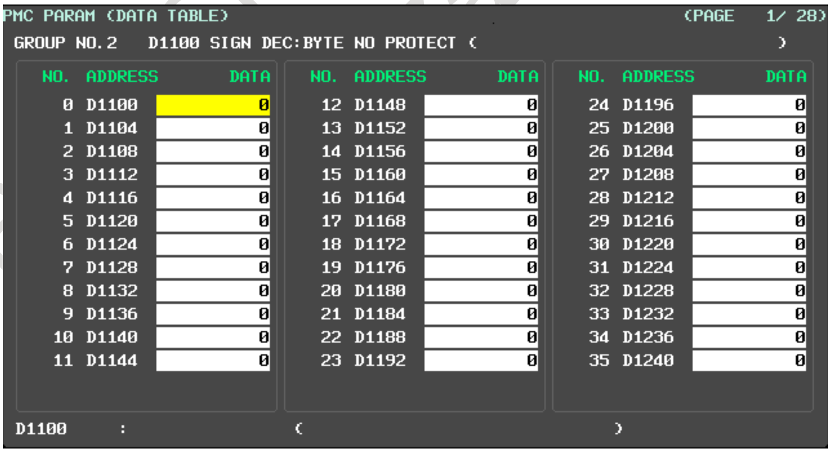
When stored stroke limit range switching is canceled, the range of stored stroke limit of the specified axis returns to the parameter setting value. Cancellation of stored stroke limit range switching consists of the next two steps.

1) Specify the kind of the stored stroke limit that is going to cancel.
   Specify the kind of stored stroke limit that is going to cancel with stored stroke limit range switching cancellation signals +/-OT11C,+/-OT12C,+/-OT2C,+/-OT3C <Gn598>. Two or more kinds of stored stroke limit can be specified at the same time.

2) Request and finish of cancellation
   When step 1) has been prepared, set stored stroke limit range switching start signal to "1".
   When stored stroke limit range switching finish signal becomes "1", the range of stored stroke limit of the specified axis returns to the parameter setting value, therefore set stored stroke limit range switching start signal to "0".

- Data format for switching

The data format is described as follows.
- Data for switching uses 4 bytes per data.
- Set the value that becomes the multiple of four (0, 4, and 8, etc.) to data table address that sets the switching data. Therefore, the total of the offset specified by the parameter No.1313 and the stored stroke limit range switching data selection signal should be a multiple of four (0, 4, and 8, etc.).
- Set data for switching to data table by a diameter value according to diameter specification of axes.
- Set data for switching to data table by detection units.
  Example)
  Data for switching is 1000 for 1.000(IS-B).
  Data for switching is 10000 for 1.0000(IS-C).
- Please set data for switching continuously, without creating a blank area. For example, please set the data of the third axis directly soon after the data of the first axis when you would like to switch the first axis and third axis.


PMC PARAM (DATA TABLE) (PAGE 1/ 28)
GROUP NO.2 D1100 SIGN DEC:BYTE NO PROTECT ( )

NO.	ADDRESS	DATA	NO.	ADDRESS	DATA	NO.	ADDRESS	DATA
0	D1100	0	12	D1148	0	24	D1196	0
1	D1104	0	13	D1152	0	25	D1200	0
2	D1108	0	14	D1156	0	26	D1204	0
3	D1112	0	15	D1160	0	27	D1208	0
4	D1116	0	16	D1164	0	28	D1212	0
5	D1120	0	17	D1168	0	29	D1216	0
6	D1124	0	18	D1172	0	30	D1220	0
7	D1128	0	19	D1176	0	31	D1224	0
8	D1132	0	20	D1180	0	32	D1228	0
9	D1136	0	21	D1184	0	33	D1232	0
10	D1140	0	22	D1188	0	34	D1236	0
11	D1144	0	23	D1192	0	35	D1240	0

D1100 : ( )

Fig.2.3.10 (a) Data format for switching

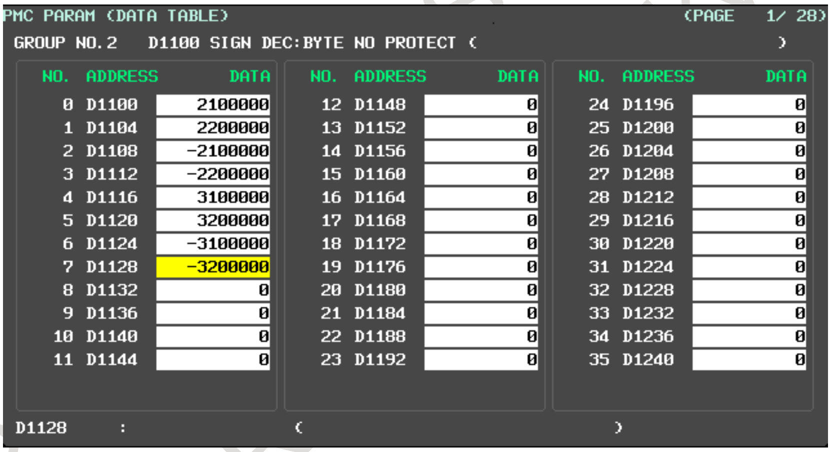
- Example of setting of data for switching and time chart

- Switching axes : The first axis and the second axis
- Switching kinds : Coordinate value of stored stroke check 2,3 in the positive direction, and the negative direction
- Head of data table (D) : Parameter No.1313=1000
- Offset from head of D : 100
- Switching data : Coordinate value of stored stroke check 2 in the positive direction
  The first axis = 2100.000
  The second axis = 2200.000
  Coordinate value of stored stroke check 2 in the negative direction

The first axis = -2100.000
The second axis = -2200.000
Coordinate value of stored stroke check 3 in the positive direction
The first axis = 3100.000
The second axis = -3200.000
Coordinate value of stored stroke check 3 in the negative direction
The first axis = -3100.000
The second axis = 3200.000

Example of setting the above-mentioned condition

Coordinate of stored stroke check 2 in the positive direction	D1100 = 2100000 -----	The first axis
	D1104 = 2200000 -----	The second axis
Coordinate of stored stroke check 2 in the negative direction	D1108 = -2100000 ----	The first axis
	D1112 = -2200000 ----	The second axis
Coordinate of stored stroke check 3 in the positive direction	D1116 = 3100000 -----	The first axis
	D1120 = 3200000 -----	The second axis
Coordinate of stored stroke check 3 in the negative direction	D1124 = -3100000 ----	The first axis
	D1128 = -3200000 ----	The second axis

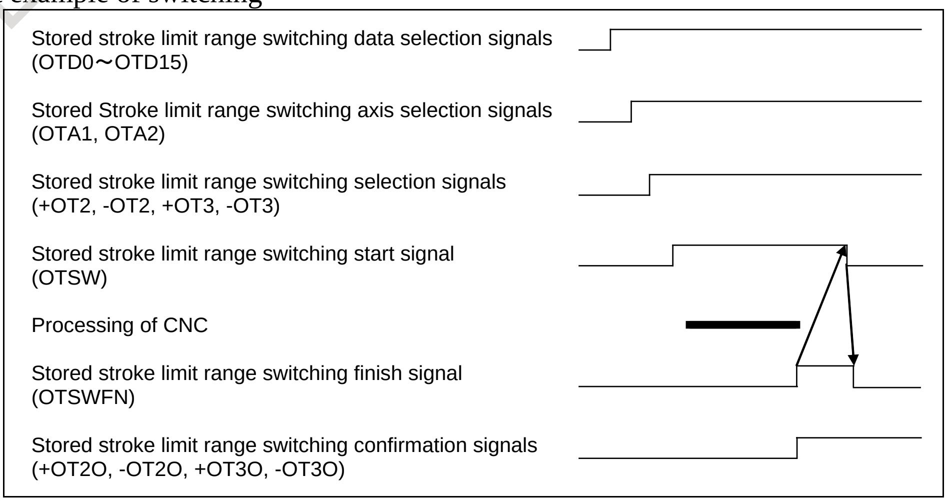

PMC PARAM (DATA TABLE) (PAGE 1/ 28)
GROUP NO.2 D1100 SIGN DEC:BYTE NO PROTECT ( )

NO.	ADDRESS	DATA	NO.	ADDRESS	DATA	NO.	ADDRESS	DATA
0	D1100	2100000	12	D1148	0	24	D1196	0
1	D1104	2200000	13	D1152	0	25	D1200	0
2	D1108	-2100000	14	D1156	0	26	D1204	0
3	D1112	-2200000	15	D1160	0	27	D1208	0
4	D1116	3100000	16	D1164	0	28	D1212	0
5	D1120	3200000	17	D1168	0	29	D1216	0
6	D1124	-3100000	18	D1172	0	30	D1220	0
7	D1128	-3200000	19	D1176	0	31	D1224	0
8	D1132	0	20	D1180	0	32	D1228	0
9	D1136	0	21	D1184	0	33	D1232	0
10	D1140	0	22	D1188	0	34	D1236	0
11	D1144	0	23	D1192	0	35	D1240	0

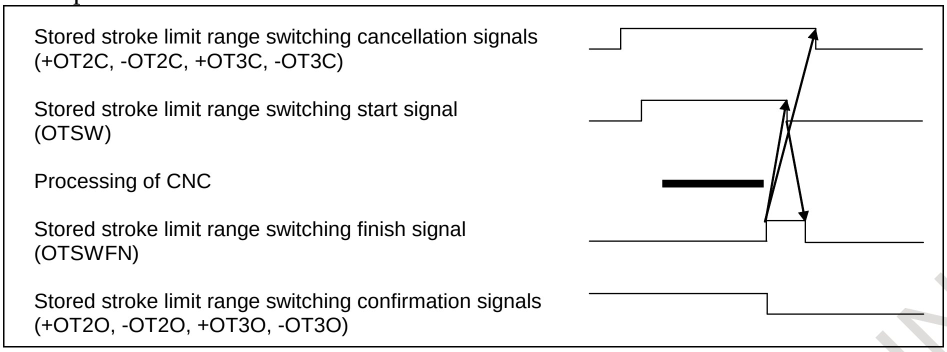
D1128 : ( )

Fig.2.3.10 (b) Example of setting data for switching

Timing chart example of switching



Timing chart example of cancellation



NOTE

1. If the offset value which is added the value specified by the parameter No.1313 and the stored stroke limit range switching data selection signal is not a multiple of four (0, 4, and 8, etc.), alarm PW0041, "DATA TABLE(D) SETTING IS ILLEGAL" is generated.
2. The area of data table (D) of PMC is different depending on PMC used and its memory size. Please confirm the specification of PMC, and set it within the range that can be used.
3. A different kind of a different axis cannot be switch at the same time though two or more kinds or two or more axes can be switched at the same time. For example, it is not possible to switch the second axis to stored stroke limit 2 and switch the first axis to stored stroke limit 3 at the same time.
4. The cancellation is given to priority when the stored stroke limit range switching start signal is set to "1" in the state of the signal that both the switching and the cancellation are possible.
5. Even if other input signals are switched in the state "1" of stored stroke limit range switching start signal, it doesn't become effective immediately. It is reflected at the time of rising of the stored stroke limit range switching start signal.
6. Confirm whether to operate correctly by data and the signal that this function set beforehand.

Signal

Stored stroke limit range switching data selection signals OTD0 to OTD15 <Gn594 to Gn595>

[Classification] Input signal

[Function] The offset value from the address of head of data table (D) of PMC to which the data for switching of stored stroke limit range switching is set is specified by the binary code in two bytes. The address of head data is set to the parameter No.1313.

[Operation] When stored stroke limit range switching start signal is set to "1", the switching operation of the range of stored stroke limit starts by the specified data.

Stored stroke limit range switching axis selection signals OTA1 to OTA8<Gn596>

[Classification] Input signal

[Function] The axis that executes stored stroke limit range switching is specified.
These signals are provided for each control axis; the affixed number of the signal name shows the control axis number.

OTAx
x : 1 ..... Stored stroke limit switching to the first axis
2 ..... Stored stroke limit switching to 2nd axis
3 ..... Stored stroke limit switching to the 3rd axis
: :

[Operation] When stored stroke limit range switching start signal is set to "1", the range of the stored stroke limit of the specified axis is switched.

Stored stroke limit range switching selection signals
+OT11, -OT11, +OT12, -OT12, +OT2, -OT2, +OT3, -OT3 <Gn597>

[Classification] Input signal

[Function] The range switching of which stored stroke limit is executed is specified.

+OT11 ..... Coordinate value I of stored stroke check 1 in the positive direction
-OT11 ..... Coordinate value I of stored stroke check 1 in the negative direction
+OT12 ..... Coordinate value II of stored stroke check 1 in the negative direction
-OT12 ..... Coordinate value II of stored stroke check 1 in the negative direction
+OT2 ..... Coordinate value of stored stroke check 2 in the positive direction
-OT2 ..... Coordinate value of stored stroke check 2 in the negative direction
+OT3 ..... Coordinate value of stored stroke check 3 in the positive direction
-OT3 ..... Coordinate value of stored stroke check 3 in the negative direction

[Operation] When stored stroke limit range switching start signal is set to "1", the range of the specified stored stroke limit is switched.

Stored stroke limit range switching cancellation signals
+OT11C, -OT11C, +OT12C, -OT12C, +OT2C, -OT2C, +OT3C, -OT3C <Gn598>

[Classification] Input signal

[Function] The range switching of which stored stroke limit is canceled is specified.

[Operation] The range of the specified stored stroke limit switches to the parameter setting value when stored stroke limit range switching start signal switches "1".

Stored stroke limit range switching start signal OTSW<Gn599.0>

[Classification] Input signal

[Function] The range switching of which stored stroke limit is started.

[Operation] When this signal is set to "1", the range of the specified stored stroke limit is switched. Or, it is canceled.
When stored stroke limit range switching finish signal is set to "0", it is accepted.

Stored stroke limit range switching confirmation signals
+OT11O, -OT11O, +OT12O, -OT12O, +OT2O, -OT2O, +OT3O, -OT3O<Fn598>

[Classification] Output signal

[Function] These signals indicate that the range of stored stroke limit specified has been switched.

[Operation] These signals are set to "1" when:
- In the same kind of stored stroke limit, the range switching has become effective even one axis.

These signals are set to "0" when:
- In the same kind of stored stroke limit, the range switching of all axes has become ineffective.

Stored stroke limit range switching finish signal OTSWFN<Fn599.0>

[Classification] Output signal

[Function] The signal indicates that stored stroke limit range switching has been completed.

[Operation] These signals are set to "1" when:
- Stored stroke limit, the range switching has been completed.

These signals are set to "0" when:

- After having completed stored stroke limit range switching, stored stroke limit range switching start signal has been become "0".

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn594	OTD7	OTD6	OTD5	OTD4	OTD3	OTD2	OTD1	OTD0
Gn595	OTD15	OTD14	OTD13	OTD12	OTD11	OTD10	OTD9	OTD8
Gn596	OTA8	OTA7	OTA6	OTA5	OTA4	OTA3	OTA2	OTA1
Gn597	-OT3	+OT3	-OT2	+OT2	-OT12	+OT12	-OT11	+OT11
Gn598	-OT3C	+OT3C	-OT2C	+OT2C	-OT12C	+OT12C	-OT11C	+OT11C
Gn599								OTSW

	#7	#6	#5	#4	#3	#2	#1	#0
Fn598	-OT3O	+OT3O	-OT2O	+OT2O	-OT12O	+OT12O	-OT11O	+OT11O
Fn599								OTSWFN

Parameter

1313	The first address of data table (D) that sets data that switches stored stroke limit

[Input type] Parameter input
[Data type] 2 word path
[Valid data range] 0 to 59996

The first address of data table (D) of PMC that sets data that switches stored stroke limit is set.

NOTE
1 1When this parameter is set, the power must be turned off before operation is continued.
2 When this parameter is 0, a data table from address D0 is used.

⚠ **WARNING**
If a data table of the set address in this parameter is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used data table.

Alarm and message

Number	Message	Description
PW0041	DATA TABLE(D) SETTING IS ILLEGAL	The address of data table (D) PMC calculated from parameter No.1313, stored stroke limit range switching data selection signal (OTD0-OTD15), and the switching data area is illegal. (1) The specified first address is illegal. (2) It is not a multiple of four. (3) The range of the specified data table is illegal.

Diagnosis data

3710	Coordinate value I of stored stroke check 1 in the positive direction
3711	Coordinate value I of stored stroke check 1 in the negative direction
3712	Coordinate value II of stored stroke check 1 in the positive direction
3713	Coordinate value II of stored stroke check 1 in the negative direction
3714	Coordinate value of stored stroke check 2 in the positive direction
3715	Coordinate value of stored stroke check 2 in the negative direction
3716	Coordinate value of stored stroke check 3 in the positive direction
3717	Coordinate value of stored stroke check 3 in the negative direction

[Data type] Real axis
[Unit of data] Machine unit

Machine coordinates value of effective, various kind of stored stroke limits are displayed now.

Note

NOTE

1. Stroke limit check before move also becomes effective in the switched area.
2. The range of stored stroke limit returns to the state set to the parameter at power on.
3. When the number of control axes in the system increases and decreases by flexible path axis assignment, it is necessary to consider it.
4. Stored stroke check 1 area expansion (coordinate value III - VIII) cannot be rewritten by this function.
5. When the range of stored stroke limit is switched by this function, even if the parameters Nos.1320 to 1327 are rewritten by the G10 instruction, the PMC window, and the G22 instruction, it is not effective immediately.
6. Stroke limit area changing function is a function to rewrite the parameters Nos.1320 to 1327. This function is a function that switches the range of stroke limit to the value that is set to data table (D) of PMC without rewriting the parameters Nos.1320 to 1327.

2.3.11 Chuck and Tail Stock Barrier

T

Overview

The chuck/tail stock barrier function prevents damage to the machine by checking whether the tool tip interferes with either the chuck or tail stock.

Specify an area into which the tool may not enter (entry-inhibition area). This is done using the special setting screen, according to the shapes of the chuck and tail stock. If the tool tip should enter the set area during a machining operation, this function stops the tool and outputs an alarm message.

The tool can be removed from the entry-inhibition area only by retracting it in the direction from which the tool entered the area.

Explanation

- Settings process for Chuck and Tail stock barrier

Refer to the "Chuck and Tail Stock Barriers" in the Operator's Manual (For Lathe System) (B-64694EN-1) for the details of setting process for Chuck and Tail stock barrier.

- Setting the entry-inhibition area for a chuck

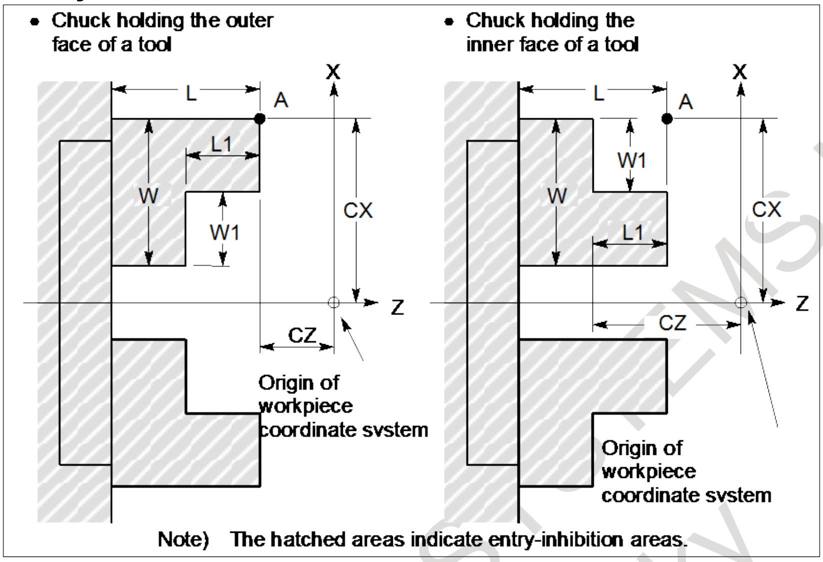


Fig. 2.3.11 (a)

Table 2.3.11 (a)

Symbol	Description	Parameter
TY	Chuck-shape selection (0: Holding the inner face of a tool, 1: Holding the outer face of a tool)	No.1330
CX	Chuck position (along X-axis)	No.1335
CZ	Chuck position (along Z-axis)	No.1336
L	Length of chuck jaws	No.1331
W	Depth of chuck jaws (radius)	No.1332
L1	Holding length of chuck jaws	No.1333
W1	Holding depth of chuck jaws (radius)	No.1334

TY : Selects a chuck type, based on its shape. Specifying 0 selects a chuck that holds the inner face of a tool. Specifying 1 selects a chuck that holds the outer face of a tool. A chuck is assumed to be symmetrical about its Z-axis.

CX, CZ : Specify the coordinates of a chuck position, point A, in the workpiece coordinate system. These coordinates are not the same as those in the machine coordinate system.

The direction of the chuck and of the tail stock is defined by a big and small relation between CZ and TZ (Z coordinate of a tail stock). Please refer to "Setting of direction of chuck and of tail stock" for details.

L, L1 : Defines the length of chuck jaws.

W, W1 : Defines the width of chuck jaws

NOTE

1. For CX and CZ, whether diameter programming or radius programming is used for the axis determines the programming system. When diameter programming is used for the axis, use diameter programming to enter data for the axis.
2. Always specify W and W1 in radius. When radius programming is used for the Z-axis, specify L and L1 in radius.
3. If you not use the entry-inhibition area for a chuck, please set 0 to L, L1, W and W1.
4. The CZ is used to define the direction of the chuck and of the tail stock. Even when you not use the entry-inhibition area for a chuck, please set to CZ.

- Setting the entry-inhibition area for a tail stock

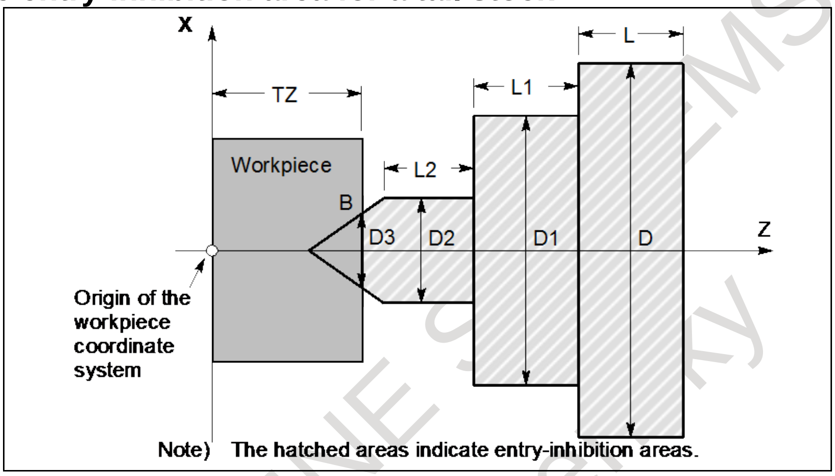


Fig. 2.3.11 (b)

Table 2.3.11 (b)

Symbol	Description	Parameter
TZ	Tailstock position (along the Z-axis)	No.1348
L	Tailstock length	No.1341
D	Tailstock diameter	No.1342
L1	Tailstock length (1)	No.1343
D1	Tailstock diameter (1)	No.1344
L2	Tailstock length (2)	No.1345
D2	Tailstock diameter (2)	No.1346
D3	Hole diameter of tailstock (3)	No.1347

TZ : Specifies the Z coordinate of the chuck position, point B (Fig. 2.3.11 (b)), in the workpiece coordinate system. These coordinates are not the same as those in the machine coordinate system. A tail stock is assumed to be symmetrical about its Z-axis. The direction of the chuck and of the tail stock is defined by a big and small relation between CZ (Z coordinate of a chuck) and TZ. Please refer to "Setting of direction of chuck and of tail stock" for details.

L, L1, L2 : Defines the tail stock length.

D, D1, D2, D3 : Defines the tail stock diameter.

NOTE
1 For TZ, whether diameter programming or radius programming is used for the Z-axis determines the programming system.
2 Always specify D, D1, D2, and D3 in diameter programming. When radius programming is used for the Z-axis, specify L, L1, and L2 in radius.
3 If you not use the entry-inhibition area for a tail stock, please set 0 to L, L1, L2, D, D1, D2 and D3.
4 The TZ is used to define the direction of the chuck and of the tail stock. Even when you not use the entry-inhibition area for a tail stock, please set to TZ.

- Setting the entry-inhibition area for the tail stock tip

The tip angle of the tail stock is 60 degrees. The entry-inhibition area is set around the tip, assuming the angle to be 90 degrees, as shown below.

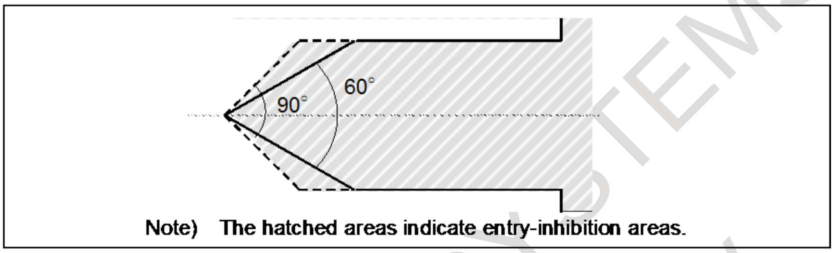


Fig. 2.3.11 (c)

- Setting of direction of chuck and of tail stock

Setting of direction of entry-inhibition area for a chuck and of entry-inhibition area for a tail stock is decided by a big and small relation between parameter No.1336 (CZ) and parameter No.1348 (TZ). When set to CZ<TZ, the direction of the chuck and of the tail stock becomes direction shown in Fig. 2.3.11 (d). When set to CZ>TZ, the direction of the chuck and of the tail stock becomes direction shown in Fig. 2.3.11 (e).

When you use only entry-inhibition area for a chuck, please set the following to parameter No.1348 (TZ).

- When you set the direction of the chuck as shown in Fig. 2.3.11 (d), please set a value that is larger than CZ to TZ
- When you set the direction of the chuck as shown in Fig. 2.3.11 (e), please set a value that is smaller than CZ to TZ

When you use only entry-inhibition area for a tail stock, please set the following to parameter No.1336 (CZ).

- When you set the direction of the tail stock as shown in Fig. 2.3.11 (d), please set a value that is smaller than TZ to CZ
- When you set the direction of the tail stock as shown in Fig. 2.3.11 (e), please set a value that is larger than TZ to CZ

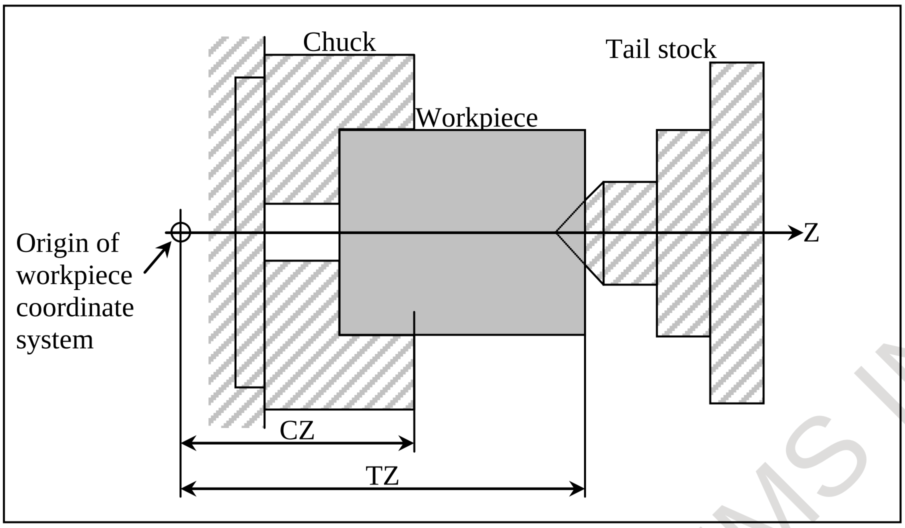


Fig. 2.3.11 (d) For parameter No.1336(CZ) < parameter No.1348(TZ)

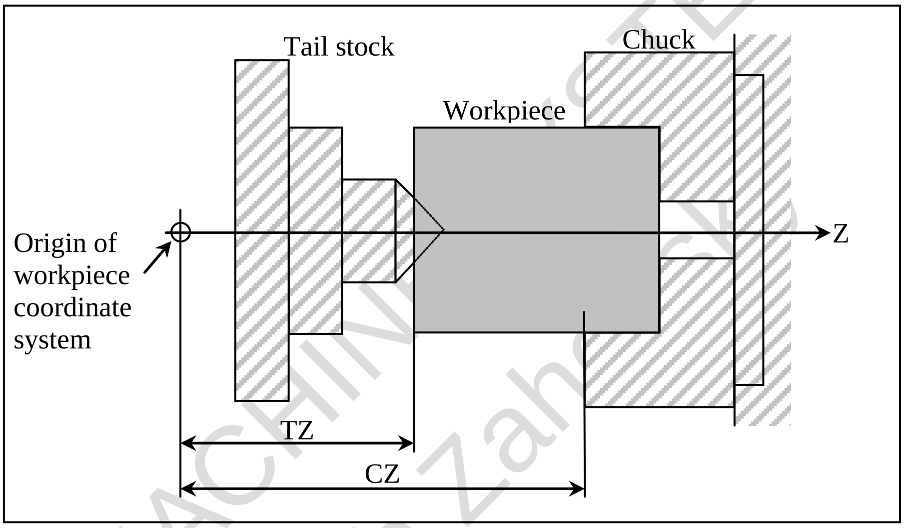


Fig. 2.3.11 (e) For parameter No.1336(CZ) > parameter No.1348(TZ)

Signal

Tailstock barrier select signal *TSB<Gn060.7>

[Classification] Input signal

[Function] Enables or disables the tail stock barrier.

[Operation] When this signal is set to “1”, the control unit operates as follows:
Disables the tail stock barrier, even when the G22 command (stored stroke check on) is specified in the program.

Table 2.3.11 (c)

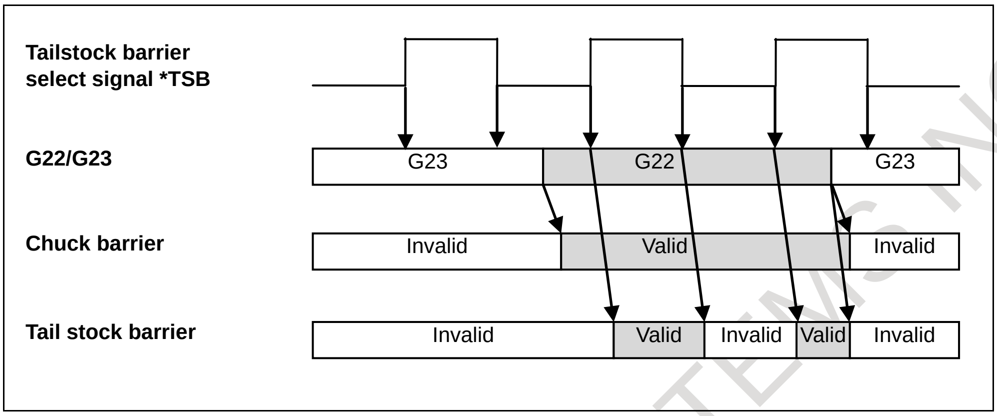
G code	*TSB	Tail stock barrier	Chuck barrier (reference)
G22	0	Enabled	Enabled
	1	Disabled	Enabled
G23	0	Disabled	Disabled
	1	Disabled	Disabled

When the G23 command (stored stroke check off) is specified, the tail stock barrier is disabled regardless of the signal *TSB. When the G22 command (stored stroke check on) is specified, the tail stock can be disabled by setting the signal to “1”.
This signal is used to select whether the tail stock area is a entry-inhibition area.
It is used whenever M commands are applied, resulting in the tail stock being attached to the workpiece or detached from the workpiece while the workpiece is being machined.

Signal address

		#7	#6	#5	#4	#3	#2	#1	#0
Gn060		*TSB							

Timing chart



Limitation

- Correct setting of an entry-inhibition area

If an entry-inhibition area is incorrectly set, it may not be possible to make the area effective. Avoid making the following settings:

- L < L1 or W < W1 in the chuck-shape settings.
- D2 < D3 in the tail stock-shape settings.
- A chuck setting overlapping that of the tail stock.

- When you use only the chuck or only the tail stock

Setting of direction of entry-inhibition area for a chuck and of entry-inhibition area for a tail stock is decided by a big and small relation between CZ and TZ. Even when you use only the chuck or only the tail stock, please set a big and small relation between CZ and TZ appropriately. Please refer to "Setting of direction of chuck and of tail stock" for details.

If you not use the entry-inhibition area for a chuck, please set 0 to L, L1, W and W1 of the chuck-shape settings.
If you not use the entry-inhibition area for a tail stock, please set 0 to L, L1, L2, D, D1, D2 and D3 of the tail stock-shape settings.

- Retraction from the entry-inhibition area

If the tool enters the entry-inhibition area and an alarm is issued, switch to manual mode, retract the tool manually, then reset the system to release the alarm. In manual mode, the tool can be moved only in the opposite direction to that in which the tool entered the area.
The tool cannot be moved in the same direction (further into the area) as it was traveling when the tool entered the area.
When the entry-inhibition areas for the chuck and tail stock are enabled, and the tool is already positioned within those areas, an alarm is issued when the tool moves.
When the tool cannot be retracted, change the setting of the entry-inhibition areas, such that the tool is outside the areas, reset the system to release the alarm, then retract the tool. Finally, reinstall the original settings.

- Coordinate system

An entry-inhibition area is defined using the workpiece coordinate system. Note the following.

<1> When the workpiece coordinate system is shifted by means of a command or operation, the entry-inhibition area is also shifted by the same amount.

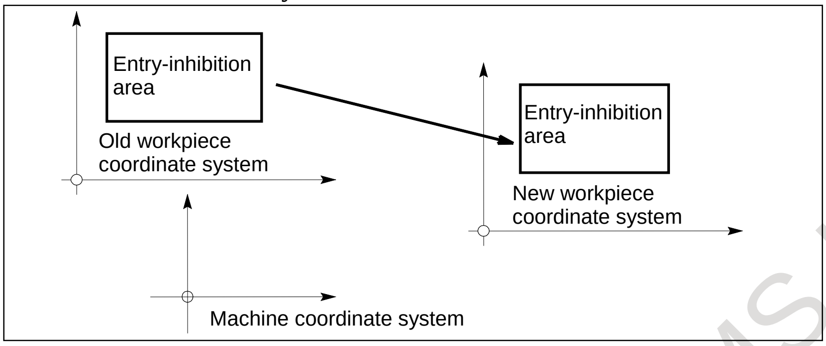


Fig. 2.3.11 (f)

Use of the following commands and operations will shift the workpiece coordinate system.
Commands:
G54 to G59, G52, G50 (G92 in G code system B or C)
Operations:
Manual handle interrupt, change in offset relative to the workpiece reference point, change in tool offset (tool geometry compensation), operation with machine lock, manual operation with manual absolute signal off

<2> When the tool enters an entry-inhibition area during automatic operation, set the manual absolute signal, *ABSM <Gn006.2>, to 0 (on), then manually retract the tool from the area. If this signal is "1", the distance the tool moves in manual operation is not counted in the tool coordinates in the workpiece coordinate system. This results in the state where the tool can never be retracted from the entry-inhibition area.

Parameter

- Profile of a chuck

1330	Profile of a chuck

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 1
Select a chuck figure.
0 : Chuck which holds a workpiece on the inner surface
1 : Chuck which holds a workpiece on the outer surface

1331	Dimensions of the claw of a chuck (L)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the length (L) of the claw of the chuck.

> **NOTE**
> Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1332	Dimensions of the claw of a chuck (W)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the width (W) of the claw of the chuck.

> **NOTE**
> Specify this parameter by using a radius value at all times.

1333	Dimensions of the claw of a chuck (L1)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the length (L1) of the claw of the chuck.

> **NOTE**
> Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1334	Dimensions of the claw of a chuck (W1)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the width (W1) of the claw of the chuck.

> **NOTE**
> Specify this parameter by using a radius value at all times.

1335	X coordinate of a chuck (CX)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the chuck position (X coordinate) in the workpiece coordinate system.

> **NOTE**
> Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1336	Z coordinate of a chuck (CZ)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the chuck position (Z coordinate) in the workpiece coordinate system.

> **NOTE**
> 1 Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.
> 2 The direction of the chuck and of the tail stock is decided by a big and small relation between this parameter and parameter No.1348 (Z coordinate of a tail stock (TZ)). Even when you not use the chuck, please set to this parameter.

- Profile of a tail stock

1341	Length of a tail stock (L)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the length (L) of the tail stock.

> **NOTE**
> Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1342	Diameter of a tail stock (D)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the diameter (D) of the tail stock.

> **NOTE**
> Specify this parameter by using a diameter value at all times.

1343	Length of a tail stock (L1)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the length (L1) of the tail stock.

> **NOTE**
> Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1344	Diameter of a tail stock (D1)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the diameter (D1) of the tail stock.

> **NOTE**
> Specify this parameter by using a diameter value at all times.

1345	Length of a tail stock (L2)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the length (L2) of the tail stock.

> **NOTE**
> Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1346	Diameter of a tail stock (D2)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the diameter (D2) of the tail stock.

> **NOTE**
> Specify this parameter by using a diameter value at all times.

1347	Diameter of a tail stock (D3)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the diameter (D3) of the tail stock.

> **NOTE**
> Specify this parameter by using a diameter value at all times.

1348	Z coordinate of a tail stock (TZ)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the tail stock position (Z coordinate) in the workpiece coordinate system.

> **NOTE**
> 1 Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.
> 2 The direction of the chuck and of the tail stock is decided by a big and small relation between this parameter and parameter No.1336 (Z coordinate of a chuck (CZ)). Even when you not use the tail stock, please set to this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
1370								CTA

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 CTA The overtravel alarm of Chuck and Tail Stock Barrier is,
0: Conventional specification. (Alarm OT0502, OT0503)
1: New specification. (Alarm OT0520, OT0521)

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23							

[Input type] Parameter input
[Data type] Bit path

#7 G23 When the power is turned on
0: G22 mode (stored stroke check on)
1: G23 mode (stored stroke check off)

Alarm and message

Number	Message	Description
OT0502	+ OVERTRAVEL (SOFT 2)	Exceeded the positive side stored stroke limit 2. Or, in the chuck tail stock barrier, an entry to the entry-inhibition area was made during movement in the positive direction.
OT0503	- OVERTRAVEL (SOFT 2)	Exceeded the negative side stored stroke limit 2. Or, in the chuck tail stock barrier, an entry to the entry-inhibition area was made during movement in the negative direction.
OT0520	+ OVERTRAVEL (CHUCK TAIL BARRIER)	In the chuck tail stock barrier, an entry to the entry-inhibition area was made during movement in the positive direction.
OT0521	- OVERTRAVEL (CHUCK TAIL BARRIER)	In the chuck tail stock barrier, an entry to the entry-inhibition area was made during movement in the negative direction.

Warning

> ⚠ **WARNING**
>
> 1 If an alarm is issued, the tool stops before it enters the entry-inhibition area when bit 7 (BFA) of parameter No. 1300 is set to 1.
> When bit 7 (BFA) of parameter No. 1300 is set to 0, an actual stop position can be in the entry-inhibition area beyond the specified profile because the CNC and machine system stop with some delay in time. For this reason, for safety, set profile data a little larger than the actual profile. The extra distance, L, required for this purpose is calculated from a rapid traverse rate as follows:
> L = (rapid-traverse-rate) $\times$ 1/7500
> For example, when a rapid traverse rate of 15 m/min is used, set profile data 2 mm larger than the actual profile.
> The profile of a chuck or tail stock can also be set using parameters Nos. 1330 to 1348.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Check/tail stock barrier

2.4 ALARM SIGNALS

When an alarm is triggered in the CNC, the alarm is displayed on the screen, and the alarm signal is set to "1".
If the voltage level of the memory backup battery falls to below a specified level while the CNC is turned off, the battery alarm signal is set to "1".

Signal

Alarm signal AL<Fn001.0>

[Classification] Output signal
[Function] The alarm signal reports that the CNC is in an alarm state.
The following are the alarms that may be issued:
(a) TH alarm
(b) TV alarm
(c) P/S alarm
(d) Overtravel alarm
(e) Overheat alarm
(f) Servo alarm
[Output cond.] The alarm signal is set to "1" when:
- The CNC is placed in the alarm state.
The alarm signal is set to "0" when:
- The alarm has been released by resetting the CNC.

Battery alarm signal BAL<Fn001.2>

[Classification] Output signal
[Function] The battery alarm signal indicates that the voltage of the battery for the memory has fallen to below a specified level while the CNC is off. In general, this signal is used to turn on an LED to notify the operator.
[Output cond.] The signal is set to "1" when:
- The battery voltage has fallen to below the specified level.
The signal is set to "0" when:
- The battery voltage has risen to the specified level or higher.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn001						BAL		AL

2.5 START LOCK / INTERLOCK

These signals disable machine movement along axes. When any of these signals is activated during movement, tool movement along the affected axis (or axes) is decelerated, then stopped.

Signal

Start lock signal STLK<Gn007.1>

[Classification] Input signal

[Function] This signal disables movement along axes during automatic operation (memory operation, DNC operation, or MDI operation).

[Operation] When the start lock signal STLK is set to “1”, a movement along an axis decelerates and stops. In this case, the automatic operation mode is maintained (with the cycle start lamp signal STL set to “1” and the feed hold lamp signal SPL set to “0”) while the movement is stopped. Therefore, when blocks containing only M, S, T, and B (second auxiliary functions) commands with no axis move commands are specified successively, the M, S, T, and B functions are executed one after another until a block specifying an axis move command is encountered. When an axis move command and M, S, T, and B functions are both specified, only the M, S, T, and B functions are sent, and a stop occurs in the automatic operation mode. Setting the start lock signal STLK to “0” restarts the operation. (Fig. 2.5 (a) and Fig. 2.5 (b)).

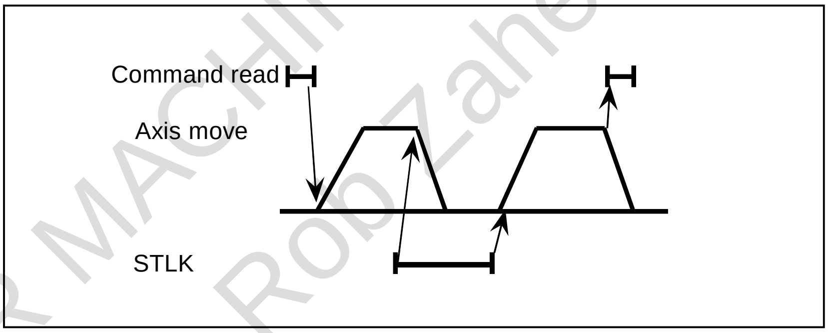


Fig. 2.5 (a) Interlock when only axis move commands are specified

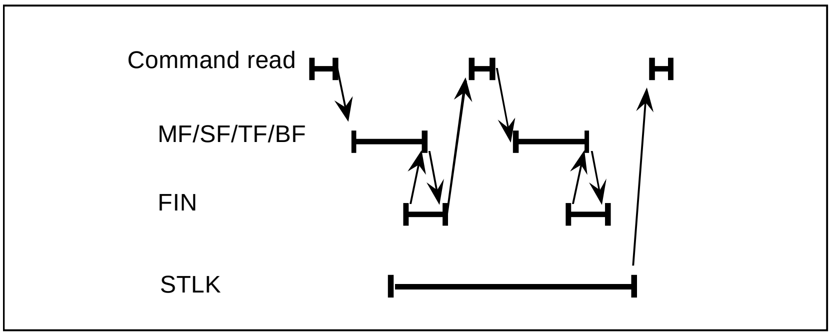


Fig. 2.5 (b) Interlock when a block containing auxiliary functions only is specified

All axes interlock signal *IT<Gn008.0>

[Classification] Input signal

[Function] This signal is used to inhibit the machine from moving, and is effective regardless of the selected mode.

[Operation] When the all axes interlock signal *IT is set to “0”, the axis movement is decelerated and stopped regardless of the selected mode. In this case, the automatic operation mode is maintained (with the cycle start lamp signal STL set to “1”and the feed hold lamp signal SPL set to “0”) while the movement is stopped. Therefore, when blocks containing only M, S, T, and B (second auxiliary functions) commands with no axis move commands are specified successively, the M, S, T, and B functions are executed one after another until a block specifying an axis move command is encountered. When an axis move command and M, S, T, and B functions are both specified, only the M, S, T, and B functions are sent, and a stop occurs in the automatic operation mode. Setting the axes interlock signal *IT to “1” restarts the operation. (Fig. 2.5 (c) and Fig. 2.5 (d)).

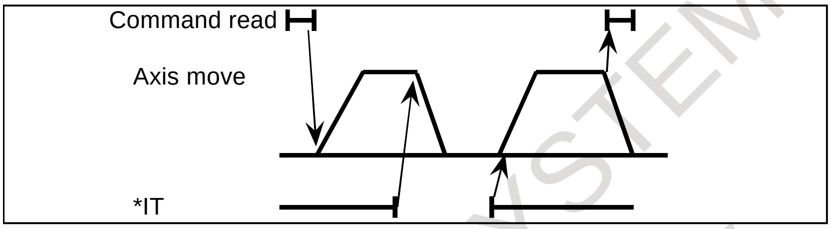


Fig. 2.5 (c) Block containing axis move commands only (manual and automatic operation)

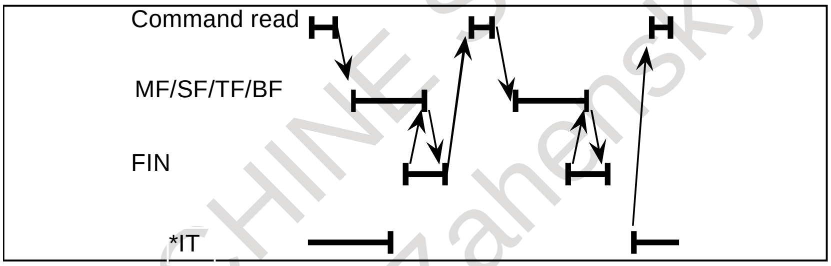


Fig. 2.5 (d) Block containing auxiliary functions only (automatic operation)

> **NOTE**
> 1 The overtravel distance of the motor after turning the all axes interlock signal *IT to “0” is represented by the following formula.
> Where
> $Qmax=Fm\times 1/60\times(A/1000+Tc/1000+Ts/1000)$
> Qmax : Overtravel quantity (mm or inch)
> Fm : Feedrate (mm/min or inch/min)
> Tc : Cutting time constant (ms)
> Ts : Servo time constant (Ts = 33ms normally)
> A : Processing time of CNC
> A = 50ms
> When acceleration/deceleration before read-ahead interpolation is used, the distance required for deceleration using look-ahead acceleration/deceleration before interpolation is added to the above formula.
> 2 Interlock signal for each axis might become invalid by the function to use. Please refer to note at this paragraph end.

Interlock signal for each axis*IT1 to *IT8<Gn130>

[Classification] Input signal

[Function] These signals disable feed along axes on an axis-by-axis basis.
A separate interlock signal is provided for each controlled axis. The number at the end of each signal name denotes the number of the corresponding controlled axis.

*IT x

x : 1 ..... Interlock for the first axis
2 ..... Interlock for the second axis
3 ..... Interlock for the third axis
: :

[Operation]
a) In manual operation
The movement of an interlocked axis is inhibited, but the other axes are movable. If an axis is interlocked during movement, it stops after being decelerated, and it starts moving again when it is released from interlock.
b) In automatic operation (MEM, RMT, or MDI mode)
If an axis is interlocked while its movement is being commanded (the move amount is not 0, inclusive of the tool offset), movement in all axes is prevented.
If a moving axis is interlocked, all axes stop moving after being decelerated, and they start moving again when interlock is released.
This function is also effective during dry run.

NOTE
Interlock signal for each axis might become invalid by the function to use. Please refer to note at this paragraph end.

Interlock signal for each axis direction
(M series) +MIT1 to +MIT8<Gn132>,-MIT1 to -MIT8<Gn134>
(T series) +MIT1<X004.2>,-MIT1<X004.3>,+MIT2<X004.4>,-MIT2<X004.5>

[Classification] Input signal

[Function] An axis direction interlock can be applied for each axis.

[Operation] When an interlock signal for each axis direction is set to “1”, the CNC applies an interlock to the specified axis direction only. However, when the interlock is done to the axis of movement instructing, the feed of all axes is prohibited regardless of axis direction.

NOTE
1 With the T series, if bit 4 (DAU) of parameter No. 3003 is set to 0, an axis direction interlock is applied only during manual operation. To apply axis direction interlock also during automatic operation, set bit 4 (DAU) of parameter No. 3003 to 1.
2 With the T series, if bit 0 (GSC) of parameter No. 5009 is set to 1, the same signal addresses as the M series (<Gn132> and <Gn134>) can be set. When using these signals in a system using four or more paths, set bit 0 (GSC) of parameter No. 5009 to 1 to use signal addresses <Gn132> and <Gn134>.
3 Interlock signal for each axis might become invalid by the function to use. Please refer to note at this paragraph end.

Block start interlock signal *BSL<Gn008.3>

[Classification] Input signal

[Function] This signal disables the start of the next block during automatic operation

[Operation] While this signal is 0, the execution of the next block during automatic operation is not started. This signal does not affect a block that has already started, and that block is continuously executed until its end. This signal does not halt automatic operation. The command in the next block is ready for execution as a valid command, so execution restarts as soon as the signal is set to “1”.

> **NOTE**
> 1 When blocks for cycle operation are internally created by a canned cycle and so on, only the first block is generally interlocked by this signal. The intermediate blocks are executed continuously even if this signal is set to “0”.
> 2 This signal is invalid to the axis movement by the real-time custom macro sentence.

Cutting block start interlock signal *CSL<Gn008.1>

[Classification] Input signal

[Function] This signal disables the start of blocks specifying move commands other than positioning during automatic operation.

[Operation] While this signal is “0”, the execution of blocks specifying movement other than positioning during automatic operation is not started. This signal does not affect a block that has already started, and that block is continuously executed until its end. This signal does not halt automatic operation. The command in the next block is ready for execution as a valid command, so execution restarts as soon as the signal is set to “1”.

[Usage] When the spindle has been specified, or when the spindle speed has been changed, this signal can be held 0 until a target spindle speed is achieved. Then, the next cutting block can be executed at the target spindle speed.

> **NOTE**
> 1 This signal is effective for any blocks including blocks for cycle operation internally created by a canned cycle and so on.
> 2 This signal is invalid to the axis movement by the real-time custom macro sentence.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn007							STLK	
Gn008					*BSL		*CSL	*IT
Gn130	*IT8	*IT7	*IT6	*IT5	*IT4	*IT3	*IT2	*IT1
Gn132	+MIT8	+MIT7	+MIT6	+MIT5	+MIT4	+MIT3	+MIT2	+MIT1
Gn134	-MIT8	-MIT7	-MIT6	-MIT5	-MIT4	-MIT3	-MIT2	-MIT1
X004			-MIT2	+MIT2	-MIT1	+MIT1		
X013			-MIT2[#2]	+MIT2[#2]	-MIT1[#2]	+MIT1[#2]		
X011			-MIT2[#3]	+MIT2[#3]	-MIT1[#3]	+MIT1[#3]		

NOTE

For the T series, the +MIT1, -MIT1, +MIT2, and -MIT2 signals are placed in X004 (path 1), X013 (path 2), and X011 (path 3) shown above when bit 2 (XSG) of parameter No. 3008 is 0. When bit 2 (XSG) of parameter No. 3008 is 1, they are placed in the X addresses set by parameter No. 3019. (The bit positions do not change.) If bit 0 (GSC) of parameter No. 5009 is set to 1, an input signal can be input from the PMC side (Gn132 (- direction), Gn134 (+ direction)). In this case, the bit positions are the same as those in the M series.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002				XIK				

[Input type] Parameter input
[Data type] Bit path

#4 XIK When bit 1 (LRP) of parameter No. 1401 is set to 0, namely, when positioning is performed using non-linear type positioning, if an interlock is applied to the machine along one of axes in positioning,

0: The machine stops moving along the axis for which the interlock is applied and continues to move along the other axes.
1: The machine stops moving along all the axes.

	#7	#6	#5	#4	#3	#2	#1	#0
3003				DAU	DIT	ITX		ITL
					DIT	ITX		ITL

[Input type] Parameter input
[Data type] Bit path

#0 ITL Interlock signal for all axes

0: Enabled
1: Disabled

#2 ITX Interlock signals for each axis

0: Enabled
1: Disabled

#3 DIT The interlock signal for each axis direction is:

0: Valid.
1: Invalid.

#4 DAU When bit 3 (DIT) of parameter No. 3003 is set to 0, the interlock signal for each axis direction is:

0: Valid only in manual operation, and invalid in automatic operation.
1: Valid in either manual operation or automatic operation.

	#7	#6	#5	#4	#3	#2	#1	#0
3004							BCY	BSL

[Input type] Parameter input
[Data type] Bit path

#0 BSL The block start interlock signal *BSL and cutting block start interlock signal *CSL are:
0: Disabled.
1: Enabled.

#1 BCY When more than one operation is performed by one block command such as a canned cycle, the block start interlock signal *BSL is:
0: Checked only at the beginning of the first cycle.
1: Checked at the beginning of every cycle.

	#7	#6	#5	#4	#3	#2	#1	#0
3008						XSG		

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#2 XSG A signal assigned to an X address is:
0: Fixed at the address.
1: Able to be reassigned to an arbitrary X address.

> **NOTE**
> When this parameter is set to 1, set parameter No. 3013, No. 3014, No. 3012, and No. 3019. If parameter No. 3013 and No. 3014 are not set, the deceleration signal for reference position return *DEC is assigned to bit 0 of <Xn000>. If parameter No. 3012 and No. 3019 are not set, the skip signal, the PMC axis control skip signal, the measurement position arrival signal, the interlock signal for each axis direction, and the tool compensation value write signal are assigned to <Xn000>.

	#7	#6	#5	#4	#3	#2	#1	#0
5009								GSC

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 GSC When the function for direct input of offset value measured B is used, an offset write input signal is input from:

0: Machine side
1: PMC side

> **NOTE**
> When the interlock function for each axis direction is enabled (when bit 3 (DIT) of parameter No. 3003 is set to 0), switching can also be made between input from the machine side and input from PMC side for the interlock function for each axis direction.

	#7	#6	#5	#4	#3	#2	#1	#0
11600						D3IT		

[Input type] Parameter input
[Data type] Bit path

#2 D3IT In the 3-dimensional coordinate system conversion mode, the valid interlock signals (interlock signal for each axis (*ITx) or interlock signal for each axis direction (MITx, PITx)) are:

0: The signals for all of the target axes for 3-dimensional coordinate system conversion.
1: The signals for axes along which a movement is made during 3-dimensional coordinate system conversion.

Note

NOTE

1 The interlock signal for each axis direction (T series) can be used regardless of whether the function for direct input of offset value measured B is provided or not.

2 When an interlock is being applied to an axis, "I" appears on the left side of the axis address of the axis on the coordinate position display screen.

3 There are the following functions to invalidate each interlock signal temporarily.
   - (1) Interlock signal is invalid to the axes engaged in the synchronization establishment of axis synchronous control.
   - (2) Interlock signal is invalid to slave axis in axis synchronous control.
   - (3) In arbitrary angular axis control, when a movement is made along the angular axis only, interlocking the perpendicular axis does not interlock a movement along the perpendicular axis made by a movement along the angular axis.
   - (4) Interlock signal is invalid to slave axis in EGB synchronization.
   - (5) All axes interlock signal and interlock signal for each axis are invalid to a spindle during arbitrary speed threading.
   - (6) Interlock signal of the slave axis is invalid to the superimposed control pulse added from the master axis during superimposed control.
   - (7) In spindle orientation, all axes interlock signal and interlock signal for each axis are checked only when a block is started. A signal is ignored if input during the execution of the block.
   - (8) Interlock signal is invalid to the axis control command of the real-time custom macro.
   - (9) In polygon turning, interlock signal is invalid to the tool rotation axis while synchronous operation.
   - (10) Interlock signals are invalid for PMC axis control.
   - (11) Interlock signal is invalid to the synchronous slave axis of synchronous control. Interlock signal is invalid to the composite side of composite control.

5 If interlock is applied to the oscillation axis during oscillation motion, the oscillation axis stops immediately.

6 In synchronous control, interlock signals on the synchronous master side are effective for the synchronous slave axes.
   In composite control, interlock signals in the specified path are effective.

7 The behavior of interlock signals in each axis or interlock signal for each axis direction during 3-dimensional coordinate system conversion mode follows the setting of bit 2 (D3IT) of parameter No.11600 and bit 5 (ITM) of parameter No.11223. For details, see the sections on "3-dimensional coordinate conversion".

2.6 MODE SELECTION

Overview

The mode selection signal is a code signal consisting of the three bits MD1, MD2, and MD4. The following five modes can be selected depending on the combination of these signals.

- Memory edit (EDIT)
- Memory operation (MEM)
- Manual data input (MDI)
- Manual handle/incremental feed (HANDLE/INC)
- Manual continuous feed (JOG)

And in addition, DNC operation mode can be selected by combining the memory operation (MEM) mode setting and the DNCI signal. Manual reference position return mode can be selected by combining the manual continuous feed (JOG) mode setting and the ZRN signal.
The currently selected operation mode can be posted by outputting the operation mode check signal.

Signals

Signals for mode selection

Select the desired mode by setting signals for mode selection listed below according to the setting shown in Table 2.6 (a).

Mode selection signals MD1,MD2,MD4<Gn043.0 to .2>

DNC operation selection signal DNCI<Gn043.5>

Signal for selecting manual reference position return ZRN<Gn043.7>

Signals for mode check

Check the current mode by reading the mode check signal(s) according to the setting shown in Table 2.6 (a).

Signal for checking incremental feed selection MINC<Fn003.0>

Signal for checking manual handle feed selection MH<Fn003.1>

Signal for checking jog feed selection MJ<Fn003.2>

Signal for checking manual data input selection MMDI<Fn003.3>

Signal for checking DNC operation selection MRMT<Fn003.4>

Signal for checking memory operation selection MMEM<Fn003.5>

Signal for checking program edit selection MEDT<Fn003.6>

Signal for checking teach-in selection MTCHIN<Fn003.7>

Signal for checking manual reference position return selection MREF<Fn004.5>

Table 2.6 (a) Mode selection signals and corresponding check signals
(A hyphen (-) indicates that the mode is not related to the status of the signal.)

Mode		Input signal					Output signal
		MD4	MD2	MD1	DNCI	ZRN	
Automatic operation	Manual data input (MDI)	0	0	0	-	-	MMDI
	Memory operation (MEM)	0	0	1	0	-	MMEM
	DNC operation (RMT)	0	0	1	1	-	MRMT
Memory edit (EDIT)		0	1	1	-	-	MEDT
Manual operation Manual	Manual handle/ incremental feed (HANDLE/INC)	1	0	0	-	-	MH,MINC
	Jog feed	1	0	1	-	0	MJ
	Manual reference position return	1	0	1	-	1	MREF
	Teach-in handle feed	1	1	1	-	-	MTCHIN
	Teach-in jog feed	1	1	0	-	-	MTCHIN

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn043	ZRN		DNCI			MD4	MD2	MD1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn003	MTCHIN	MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC
Fn004			MREF					

Note

Precautions on modes and mode switching

NOTE

1 In MDI mode, the cycle start lamp signal STL turns to “0” and the CNC stops as soon as the commands entered via the MDI have been executed. But the feed hold lamp signal SPL does not turn to “1”. Therefore, another command can be entered from the manual data input unit under this condition.

2 Manual operation in jog feed mode
   (1) When bit 0 (JHD) of parameter No. 7100 is set to 0
       Only jog feed is possible.
   (2) When bit 0 (JHD) of parameter No. 7100 is set to 1
       Both jog feed and manual handle feed are possible, provided the manual handle feed function is installed.

3 Manual operation in manual handle/incremental feed mode.
   (1) When the manual handle feed function is not used
       Incremental feed is valid.
   (2) When the manual handle feed function is used and bit 0 (JHD) of parameter No. 7100 is set to 0
       Only manual handle feed is valid.
   (3) When the manual handle feed function is used and bit 0 (JHD) of parameter No. 7100 is set to 1
       Incremental feed and manual handle feed are valid.

4 When switching to manual data input mode is made during operation in memory operation mode, the CNC enters the automatic operation stop state after executing the command in the current block. Cycle start lamp signal STL is then set to “0”.
   In this case, feed hold lamp signal SPL is not set to “1” (Fig. 2.6 (a)).
   When switching to memory operation mode is made during operation in manual data input mode, the CNC enters memory operation mode after executing the currently executed command. To perform memory operation at this time, the program for MDI operation must have been executed (Fig. 2.6 (b))

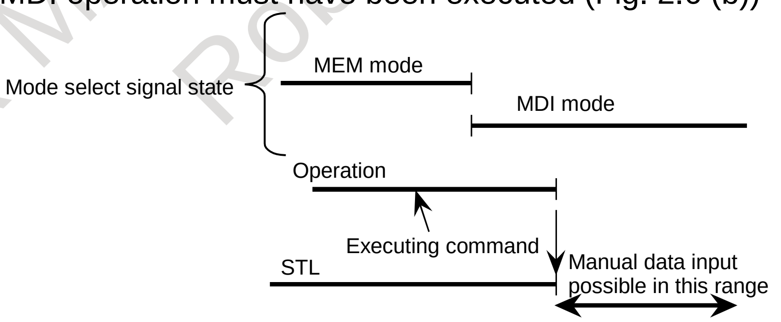


Fig. 2.6 (a)

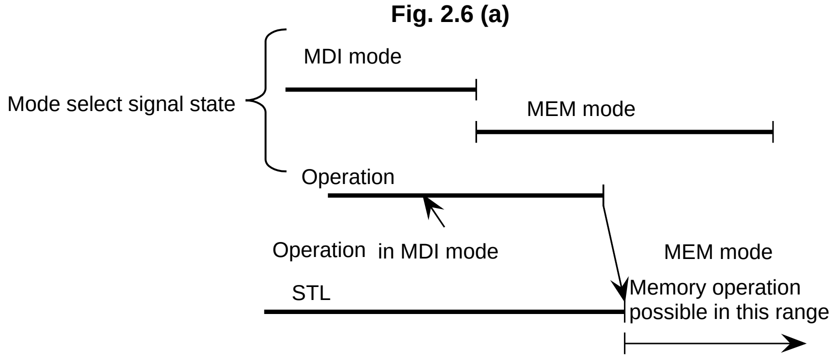


Fig. 2.6 (b)

NOTE

5 When the HANDLE/INC mode is selected while the CNC is operating in the MEM, RMT or MDI mode, the automatic or MDI operation stops, the Cycle time lamp signal STL turns to “0”, the feed hold lamp signal SPL simultaneously turns to “1”, and the CNC enters the HANDLE/INC mode. Under these conditions, manual handle feed or incremental feed by axis direction select signal is permitted. Since the MEM, RMT or MDI mode commands are held, operation can be restarted by the cycle start signal and by selecting the MDI, RMT or MEM mode. However, if operation was stopped by switching to the HANDLE/INC mode during manual data input or during automatic operation, it can be restarted only by reactivating the mode in use before the operation was stopped (Fig. 2.6 (c)).

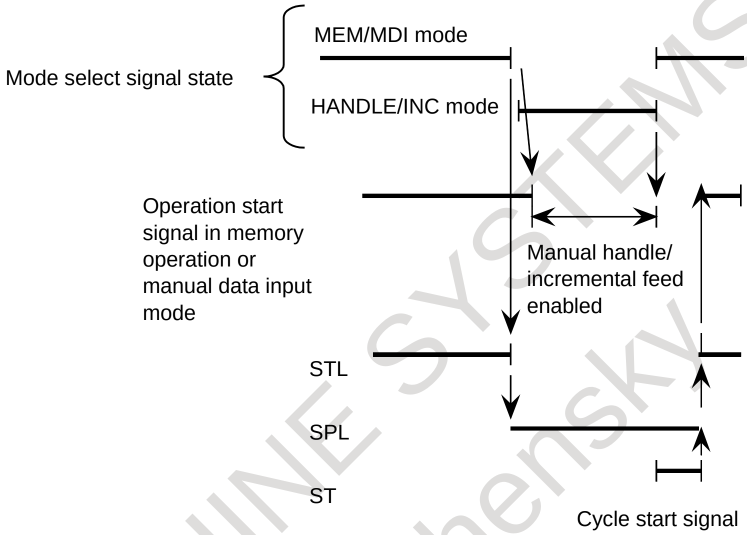


Fig. 2.6 (c)

NOTE

6 When the JOG mode is selected during MEM, RMT or MDI mode operation, operation stops, the cycle time lamp signal STL turns to “0”, the feed hold lamp signal SPL simultaneously turns to “1”, and the CNC enters the JOG mode. Under these conditions, manual feed by feed axis direction select signal is permitted. Operation can be restarted by returning to the original mode, as described for HANDLE/INC mode (Fig. 2.6 (d)). When the mode is switched to the JOG mode during manual handle feed or during incremental feed operation, the CNC ignores the manual handle feed or incremental feed command and manual jog feed becomes effective. If a feed axis direction select signal turns to 1 before the JOG mode is selected, that signal is ignored. The feed axis select signal is selected by turning the necessary feed axis direction signal to “1” after turning all the feed axis direction select signals to “0” (Fig. 2.6 (e)).
It is possible to perform handle feed in JOG mode by activating bit 0 (JHD) of parameter No. 7100. For details, refer to Note 2.

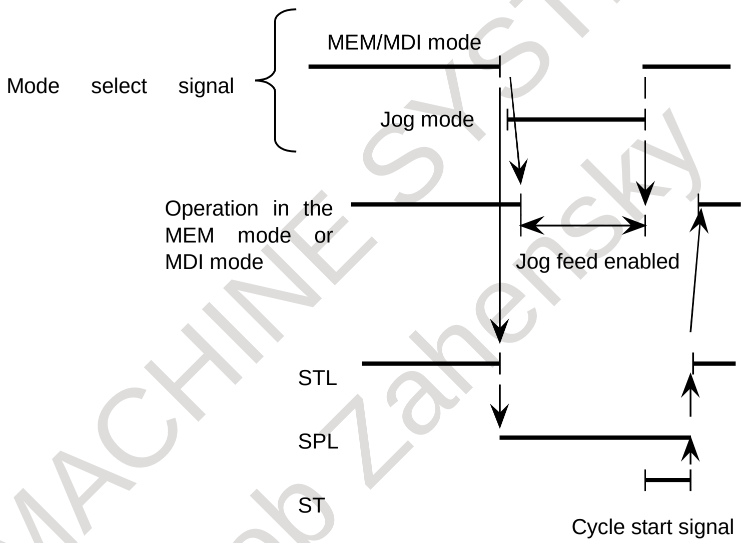


Fig. 2.6 (d)

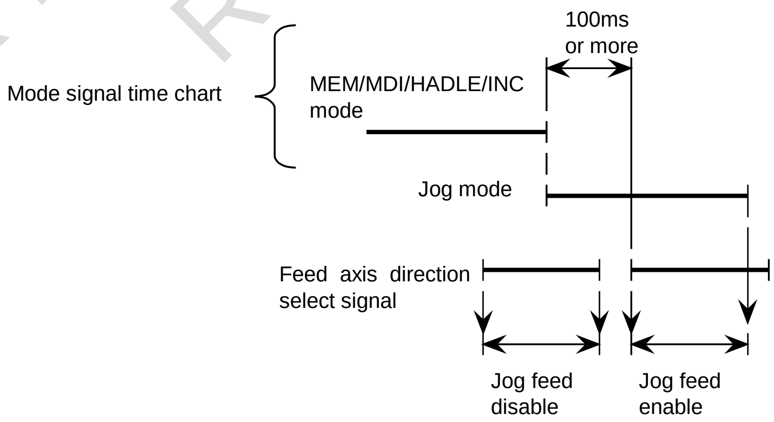


Fig. 2.6 (e)

NOTE

7 The mode switching operation is summarized in the time chart below (Fig. 2.6 (f)).

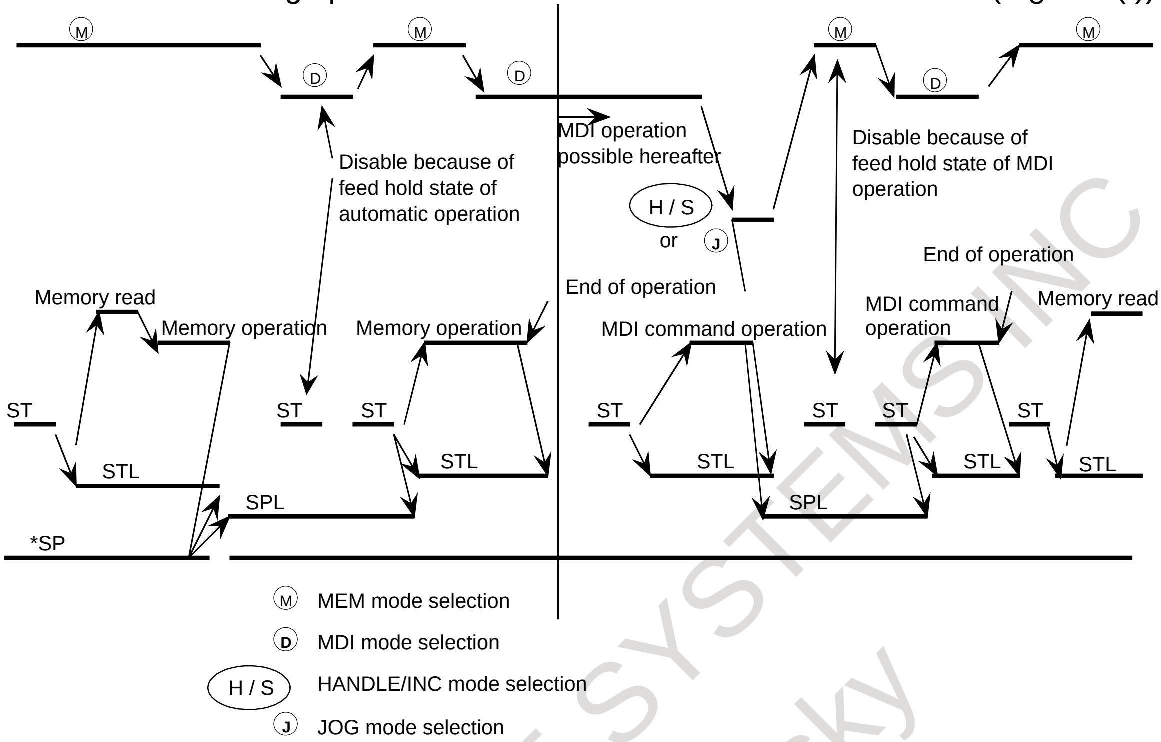


Fig. 2.6 (f) Mode signal time chart

8 Invalid mode selection signals
When mode selection signals MD4, MD2, and MD1 are set to “0”, “1”, and “0”, respectively, the mode before the signals are changed is selected. When mode selection signals MD4, MD2, and MD1 have been set to “0”, “1”, and “0”, respectively, since power-on, the MDI mode is selected.

9 Notes on output signals:
When bit 0 (JHD) of parameter No. 7100 is set to 1:
(1) In the jog feed mode, the output signals are set as follows:
MJ=”1”, MH=”0”
(2) In the manual handle feed mode, the output signals are set as follows:
MH=”1”, MINC=”0”

10 Teach-in handle/Jog mode selection output signal
(1) In teach-in handle feed mode (mode selection signals MD4, MD2, and MD1 are “1”, ”1”, and “1”), signal for checking teach-in selection MTCHIN<Fn003.7> and signal for checking manual handle feed selection MH<Fn003.1> turn to "1".
The status displayed on the bottom right on screen changes to "THND".
(2) In teach-in jog feed mode (mode selection signals MD4, MD2, and MD1 are “1”, “1”, and “0”), signal for checking teach-in selection MTCHIN<Fn003.7> and signal for checking jog feed selection MJ<Fn003.2> turn to "1".
The status displayed on the bottom right on screen changes to "TJOG".

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7100								JHD

[Input type] Parameter input
[Data type] Bit path

#0 JHD Manual handle feed in JOG feed mode or incremental feed in the manual handle feed is:
0: Invalid.
1: Valid.

2.7 STATUS OUTPUT SIGNAL

The table below lists the status output signals. They indicate the state of the CNC. See the sections listed in the Table 2.7 (a) for details of each signal.

Table 2.7 (a)

Signal name	Symbol	Reference section
Alarm signal	AL	Alarm signals
Battery alarm signal	BAL	Alarm signals
Reset signal	RST	Reset & rewind
Rewinding signal	RWD	Reset & rewind
Tapping signal	TAP	Canned cycle for drilling
Moving signals	MV1 to MV8	Axis moving status output
Moving direction signals	MVD1 to MVD8	Axis moving status output
In-position signals	INP1 to INP8	In-position check
Rapid traversing signal	RPDO	This section
Cutting feed signal	CUT	This section
Threading signal	THRD	Threading
Constant surface speed signal	CSS	Constant surface speed control
Inch input signal	INCH	Inch/metric switching
Dwell status signal	DWL	This section

Signal

Rapid traversing signal RPDO<Fn002.1>

[Classification] Output signal
[Function] This signal indicates that a move command is being executed at rapid traverse.
[Output cond.] "1" indicates an axis starts moving after rapid traverse has been selected; 0 indicates that an axis starts moving after a feedrate other than rapid traverse has been selected. This holds true for both automatic and manual operation modes.

NOTE
1. Rapid traverse in automatic operation includes all rapid traverses in canned cycle positioning, automatic reference position return, etc., as well as the move command G00. Rapid traverse in manual operation also includes rapid traverse in reference position return.
2. Once rapid traverse has been selected, this signal remains "1", including during a stop, until another feedrate has been selected and movement has been started.

Cutting feed signal CUT<Fn002.6>

[Classification] Output signal

[Function] Signals that cutting feed is being performed by automatic operation.

[Output cond.] This signal is set to “1” in the following case:

- When cutting feed is being performed by automatic operation (cutting feed for linear interpolation, circular interpolation, helical interpolation, thread cutting, skip cutting, or cutting in canned cycle)

> ⚠ **CAUTION**
> This signal is not set to “1” in the feed hold state.

> **NOTE**
> This signal is set to “1” even when the feedrate override is 0%, and even during interlock.

Dwell status signal DWL <Fn526.5>

[Classification] Output signal

[Function] This signal indicates that dwell (G04) is in progress at the automatic operation.

[Output cond.] This signal is set to “1” in the following case:

- When dwell (G04) is being performed

This signal is set to 0 in the following case.

- When dwell (G04) is not performed

When bit 2 (DWS) of parameter No.10351 is 1, dwell status signal DWL <Fn526.5> can be used.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn002		CUT					RPDO	
Fn526			DWL					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
10351						DWS		

[Input type] Parameter input

[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#2 DWS Dwell status signal DWL <Fn526.5> is:

0: Disabled.

1: Enabled.

2.8 VRDY OFF ALARM IGNORE SIGNAL

Overview

The German VDE safety standard requires that the motor be deactivated when the safety guard is opened. By using the VRDY OFF Alarm Ignore signal, however, the CNC can be restarted without resetting, even if the safety guard has been opened.

Signal

All-axis VRDY OFF alarm ignore signal IGNVRY<Gn066.0>

[Classification] Input signal

[Function] Disables the detection of servo alarm SV0401, "IMPROPER V_READY OFF", for all axes.

[Operation] When this signal is set to logical "1", the control unit operates as follows:

- The control unit does not issue servo alarm SV0401, "IMPROPER V_READY OFF", even when the servo amplifier ready signal goes off. The control unit, however, sets servo ready signal SA to "0". The SA signal can remain set to "1", depending on the setting of bit 6 (SAK) of parameter No. 1804.

Each-axis VRDY OFF alarm ignore signal IGVRY1 to IGVRY8<Gn192>

[Classification] Input signal

[Function] Disables the detection of servo alarm SV0401, "IMPROPER V_READY OFF", for the corresponding axis. These signals correspond to the controlled axes. The suffixed number of each signal corresponds to the number of the controlled axis.

[Operation] When this signal is set to logical "1", the control unit operates as follows:

- The control unit does not issue servo alarm SV0401, "IMPROPER V_READY OFF", even when the servo amplifier ready signal for the corresponding axis goes off. The servo ready signal SA, however, is set to "0". The SA signal, however, can remain set to "1" depending on the setting of bit 6 (SAK) of parameter No. 1804.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn066								IGNVRY
Gn192	IGVRY8	IGVRY7	IGVRY6	IGVRY5	IGVRY4	IGVRY3	IGVRY2	IGVRY1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1804		SAK		IVO				

[Input type] Parameter input
[Data type] Bit path

#4 IVO When an attempt is made to release an emergency stop while the VRDY OFF alarm ignore signal is "1":

0: The emergency stop state is not released until the VRDY OFF alarm ignore signal is set to "0".

1: The emergency stop state is released.

> **NOTE**
> When a reset is issued while the VRDY OFF alarm ignore signal is set to 1 and the motor activating current is low, the reset state can also be released, provided this parameter is set to 1.

#6 SAK When the VRDY OFF alarm ignore signal IGNVRY is "1", or when the VRDY OFF alarm ignore signals IGNVRYn are 1:

0: Servo ready signal SA is set to "0".

1: Servo ready signal SA remains set to "1".

Alarm and message

Number	Message	Description
SV0401	IMPROPER V_READY OFF	Although the ready signal (PRDY) of the position control was ON, the ready signal (VRDY) of the velocity control was OFF.

Caution

⚠ **CAUTION**

1. When the VRDY OFF alarm ignore signal IGNVRY is "1" and the control enters NOT READY status due to emergency stop or a servo alarm and then the control is reset, reset processing is not terminated until the VRDY OFF alarm ignore signal is set to "0".
2. When the VRDY OFF alarm ignore signal is set to "1" and the servo amplifier ready signal is set to off, the motor is freed from the drive, but follow up is not performed. To perform follow up, set the servo off signal SVFx to "1".

Note

NOTE

While the VRDY OFF alarm ignore signal is set to "1", and a servo alarm other than alarm SV0401 occurs, the control unit detects the alarm.

2.9 UNEXPECTED DISTURBANCE TORQUE DETECTION FUNCTION

Overview

Machine collision, defective, and damaged cutters cause a large load torque on the servo and spindle motors, compared with normal rapid traverse or cutting feed. This function detects the disturbance torque on the motors and sends this value as an estimated load torque to the PMC. If the detected disturbance torque value is abnormally great compared with the value specified in the parameter, the function stops the servo/spindle motor as early as possible or reverses the motor by an appropriate value specified in a parameter, in order to minimize possible damage to the machine. (The function to reverse motors is effective only for servo motors.)

Explanation

The unexpected disturbance torque detection function is further divided as follows:

(1) Estimated disturbance torque output function
The CNC is always calculating the estimated disturbance torque for the motor (excluding acceleration/deceleration torque). The estimated disturbance torque output function enables the PMC to read the calculated torque using the window function.

(2) Unexpected disturbance torque detection alarm function
This function stops motors or reverses them by an amount specified in a parameter, causing the CNC to output an alarm, whenever the disturbance torque is greater than the value specified in a parameter. (The function to reverse motors is effective only for servo motors.)]
This function can be used for the trigger condition of recording CNC information by Machine state monitoring function.

(3) Unexpected disturbance torque detection group function
This function allows you to divide servo axes into desired groups. When the disturbance torque obtained by the estimated disturbance torque output function is greater than a parameter-set value, this function stops motors immediately. The function immediately stops all axes that are set for the same group by parameter setting (including axes of which group number set to 0) then places them in the interlock state. When a value is set in parameter No. 2103, this function reverses the motors by the set amount in a direction opposite to the advance direction, then places all the axes of the same group in the interlock state.

NOTE

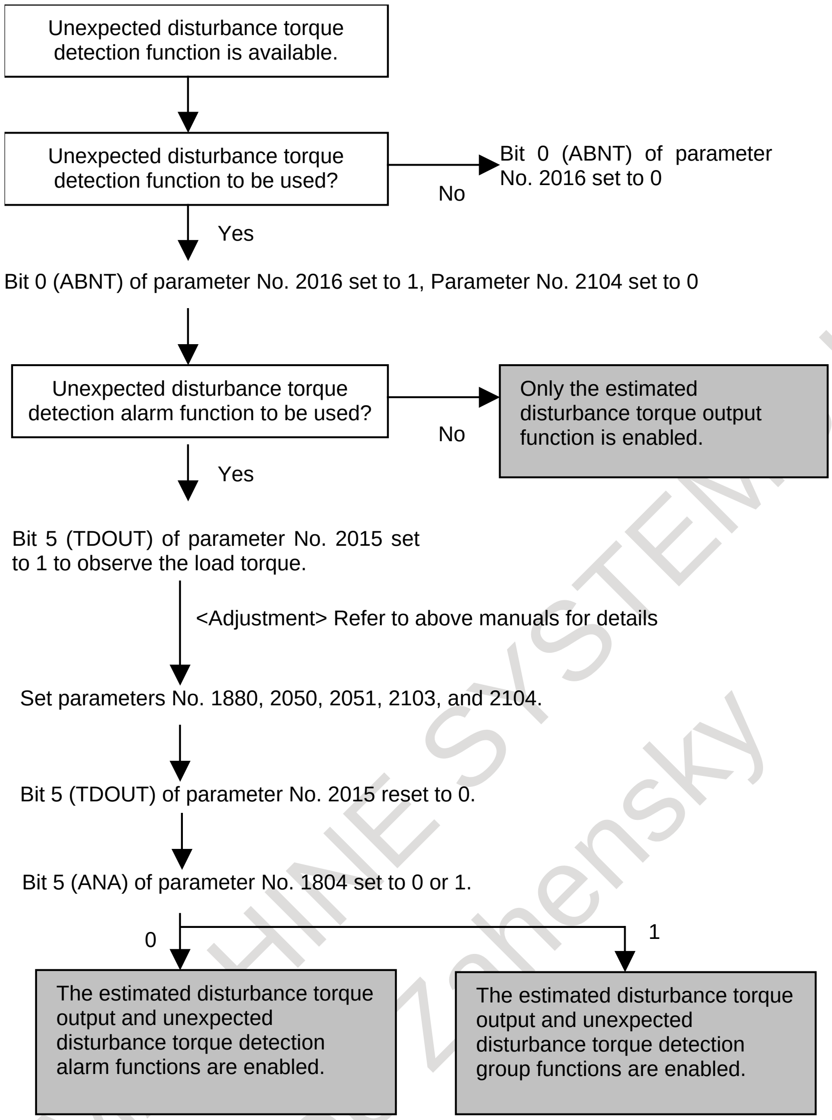
Select the unexpected disturbance torque detection alarm function or unexpected disturbance torque detection group function by setting bit 5 (ANA) of parameter No. 1804.
You cannot use these functions at the same time.

Unexpected disturbance torque detection can also be disabled only for specific axes by using bit 5 (ABDSW) of parameter No. 2215 for the unexpected disturbance torque detection function and unexpected disturbance torque detection ignore signals IUDD1 to IUDD8 <Gn125>. (This function is effective only for servo motors.)

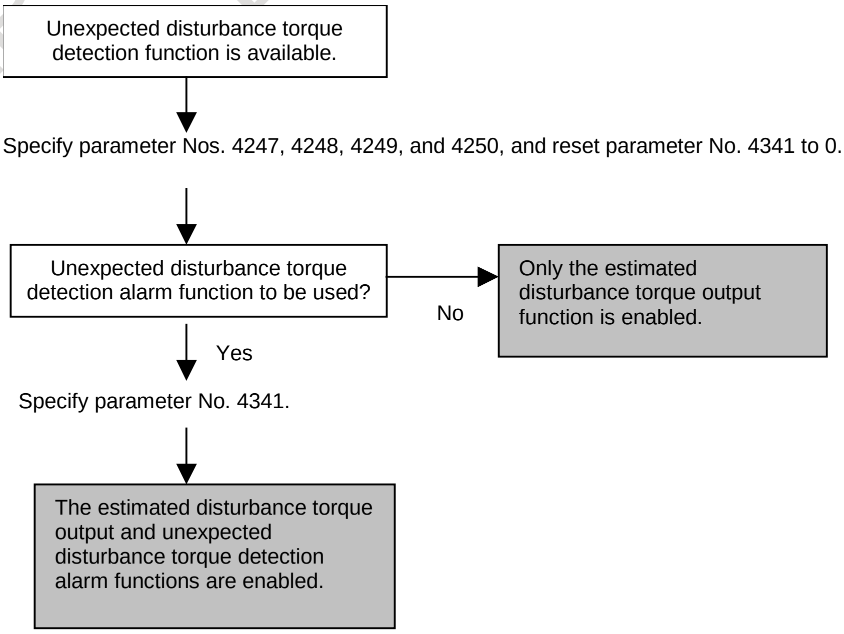
Procedure for parameter setting

The following flowcharts explain how to specify parameters for the unexpected disturbance torque detection function.
Refer to following manuals for details :
FANUC AC SERVO MOTOR α*i* series PARAMETER MANUAL B-65270EN
FANUC AC SPINDLE MOTOR α*i* series PARAMETER MANUAL B-65280EN

(1) Servo axis



(2) Spindle



- Timing chart when the unexpected disturbance torque detection group function is used

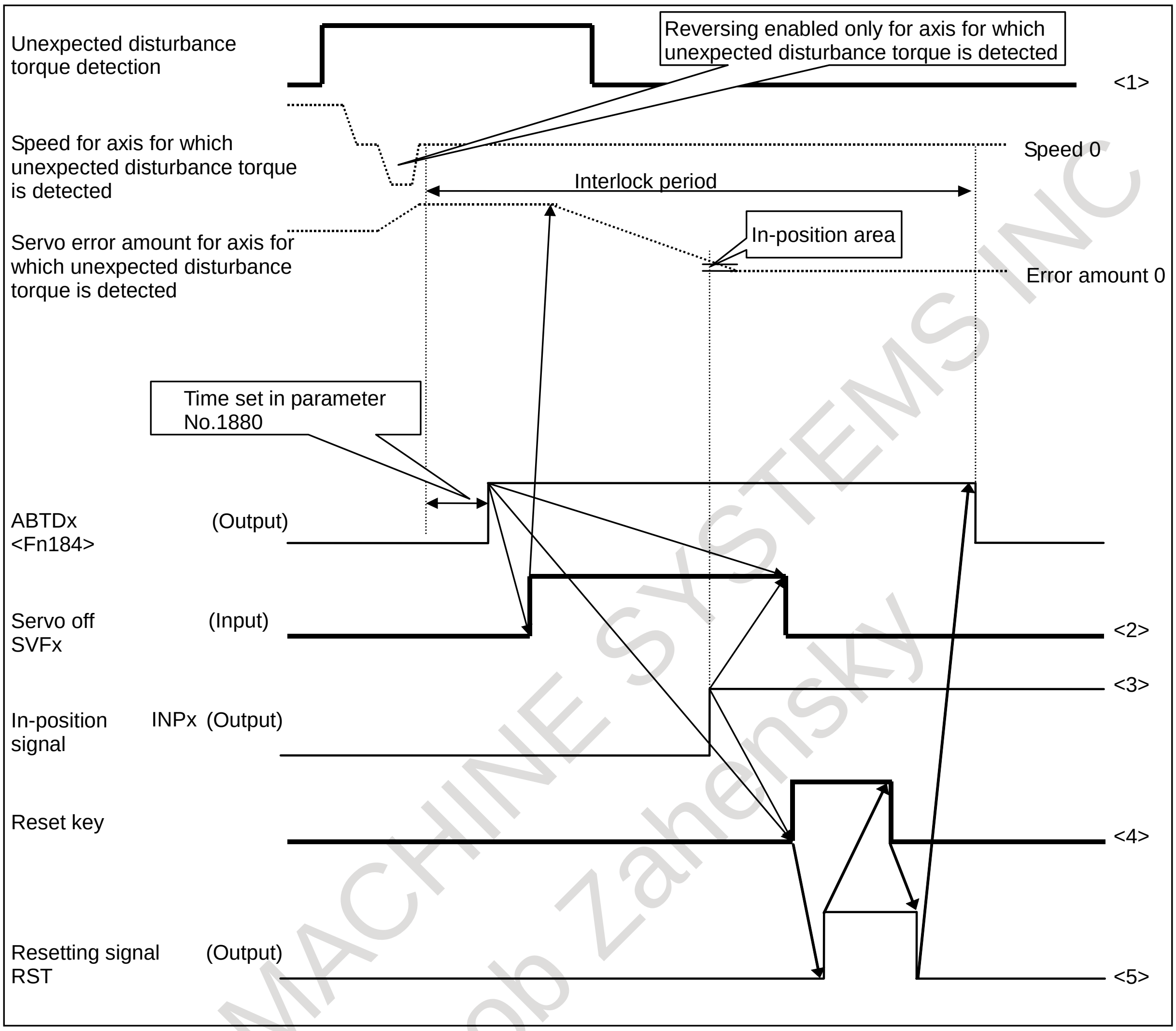


Fig. 2.9 (a) Timing chart of the unexpected disturbance torque detection group function

<1> When an unexpected disturbance torque is detected, servo motors are stopped immediately. Axes with the same group number set in parameter No. 1881 and axes with 0 set in the parameter are all stopped immediately then placed in the interlock state. At this time, axis movements are stopped regardless of commands of the CNC, so there is a difference between the coordinates held by the CNC and the actual motor position. This difference is accumulated as a servo error amount.

<2> When the operation is restarted, a follow-up operation is performed by inputting the servo off signal SVFx to match the coordinates held by the CNC and the actual motor position. Input the servo off signals SVFx for all axes of the same group (the axes with the same group number set and axes with group number 0 set). When servo off signals SVFx are input, the follow up signal *FLWU <Gn007.5> must be set to “0”, and bit 0 (FUPx) of parameter No. 1819 must be set to 0.

<3> After input servo off signal SVFx, in-position signal INPx <Fn104> becomes “1” on when the error amount enters the in-position width.

<4> Check that the in-position signals INPx for all axes that belong to the same group have became “1”, then input a reset (ERS or a reset key).

> ⚠ **CAUTION**
> When an unexpected disturbance torque is detected using the unexpected disturbance torque detection group function, be sure to follow the timing chart shown above to input servo off signals SVFx then perform a reset operation. If a reset is made without inputting the servo off signals SVFx, a shock may apply to the machine.

> **NOTE**
> 1 The servo off signal SVFx must be input with follow up signal *FLWU <Gn007.5> set to “0” and bit 0 (FUPx) of parameter No. 1819 set to 0.
> 2 Input the servo off signals SVFx for all axes of the group to which the axis with an unexpected disturbance torque detected belongs. Also for axes for which 0 is set in parameter No. 1881, input the corresponding servo off signals SVFx.
> 3 When an unexpected disturbance torque is detected for a PMC axis or when an axis for which an unexpected disturbance torque is detected belongs to a group that includes a PMC axis, all axes of the cluster to which that PMC axis belongs are placed in the interlock state.
> 4 When an unexpected disturbance torque is detected for a spindle (serial spindle), ABTSP1 <Fn090.1> or ABTSP2 <Fn090.2> is output immediately. The setting of parameter No. 1880 is ignored.
> 5 When an unexpected disturbance torque is detected for a spindle (serial spindle), servo axes are not stopped. When the servo axes must be stopped, stop them by using ladder.

Signal

Servo axis unexpected disturbance torque detection signal ABTQSV <Fn090.0>

[Classification] Output signal

[Function] Informs the PMC that an unexpected disturbance torque was detected on a servo axis.

[Output cond.] This signal becomes “1” if:

- An Unexpected disturbance torque is detected for a servo axis, Cs axis, spindle positioning axis, or spindle axis during rigid tapping.

First-spindle unexpected disturbance torque detection signal ABTSP1 <Fn090.1>

[Classification] Output signal

[Function] Informs the PMC that an unexpected disturbance torque was detected on the first spindle axis.

[Output cond.] This signal becomes “1” if:

- An unexpected disturbance torque is detected for the first spindle under speed control.

Second-spindle unexpected disturbance torque detection signal ABTSP2 <Fn090.2>

[Classification] Output signal

[Function] Informs the PMC that an unexpected disturbance torque was detected on the second spindle axis.

[Output cond.] This signal becomes “1” if:

- An unexpected disturbance torque is detected for the second spindle under speed control.

Third-spindle unexpected disturbance torque detection signal ABTSP3 <Fn090.3>

[Classification] Output signal

[Function] Informs the PMC that an unexpected disturbance torque was detected on the third spindle axis.

[Output cond.] This signal becomes “1” if:

- An unexpected disturbance torque is detected for the third spindle under speed control.

Fourth-spindle unexpected disturbance torque detection signal ABTSP4 <Fn091.4>

[Classification] Output signal

[Function] Informs the PMC that an unexpected disturbance torque was detected on the fourth spindle axis.

[Output cond.] This signal becomes “1” if:

- An unexpected disturbance torque is detected for the fourth spindle under speed control.

Unexpected disturbance torque detection signals ABDT1 to ABDT8 <Fn184>

[Classification] Output signal

[Function] Notifies the PMC of the axis on which an unexpected disturbance torque was detected. The number at the end indicates the control axis number.

[Output cond.] This signal becomes “1” if:

- An unexpected disturbance torque is detected. (At this time, the servo axis unexpected disturbance torque detection signal ABTQSV<Fn090.0> also becomes “1”.)

The Table 2.9 (a) summarizes the alarms and signals output by each function.

Table 2.9 (a)

	Signal output		Alarm	
	ABTQSV	ABTSP1 ABTSP2 ABTSP3 ABTSP4	SV0409	SP0754
Servo axis	O	-	O	-
Cs contour control	O	-	O	-
Spindle positioning axis	O	-	O	-
Rigid tapping	O	-	-	O
Spindle axis for speed control	-	O	-	O

Unexpected disturbance torque detection ignore signal IUDD1 to IUDD8<Gn125>

[Classification] Input signal

[Function] Disables the unexpected disturbance torque detection function for each axis. This signal is provided for each control axis and the value at the end indicates the control axis number.

[Output cond.] If this signal is “1”, the unexpected disturbance torque of the target axis is not detected. Before using this signal, set bit 5 (ABDSW) of the parameter No. 2215, used for the unexpected disturbance torque detection function, to 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn090					ABTSP3	ABTSP2	ABTSP1	ABTQSV
Fn091				ABTSP4				
Fn184	ABDT8	ABDT7	ABDT6	ABDT5	ABDT4	ABDT3	ABDT2	ABDT1

	#7	#6	#5	#4	#3	#2	#1	#0
Gn125	IUDD8	IUDD7	IUDD6	IUDD5	IUDD4	IUDD3	IUDD2	IUDD1

Parameters

- Common to servo axis and spindle

	#7	#6	#5	#4	#3	#2	#1	#0
1804			ANA					

[Input type] Parameter input
[Data type] Bit path

#5 ANA When an unexpected disturbance torque is detected for an axis:
0: Movement along all axes is stopped, and a servo alarm is output.
1: No servo alarm is output, and movement along only the axes of the group containing the axis with the unexpected disturbance torque is stopped in interlock mode. Each unexpected disturbance torque detection signal is output. The unexpected disturbance torque detection signal that has been output is cleared by a reset. The group number of each axis is set in parameter No.1881.

	#7	#6	#5	#4	#3	#2	#1	#0
1805				TSM	TSA			

[Input type] Parameter input
[Data type] Bit path

#3 TSA As the unexpected disturbance torque detection level during dwell, M code execution, and automatic operation halt state:
0: The threshold value for rapid traverse is used. (parameter No. 2142)
1: The threshold value for cutting feed is used. (parameter No. 2104)
If this parameter is set to 1, bit 3 (ABG0) of parameter No.2200 and bit 7 (ABT) of parameters No.2215 need to be set to 1.

#4 TSM As the unexpected disturbance torque detection level in the jog feed mode (excluding manual rapid traverse) and manual handle feed mode:
0: The threshold value for rapid traverse is used. (parameter No. 2142)
1: The threshold value for cutting feed is used. (parameter No. 2104)
This parameter is valid when bit 3 (ABG0) of parameter No. 2200 is set to 1.

1880	Abnormal load detection alarm timer

[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 32767
This parameter sets the time from the detection of an unexpected disturbance torque until a servo alarm is issued.
When 0 is set, however, the specification of 200 msec is assumed.

1881	Group number when an unexpected disturbance torque is detected

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 32
Set the group number on each axis when an unexpected disturbance torque is detected.

When an unexpected disturbance torque is detected on an axis, only the movements on those axes that belong to the same group as the axis are stopped.
If 0 is set for an axis, the movement on the axis is stopped when an unexpected disturbance torque is detected on any other axis.
This parameter is valid when bit 5 (ANA) of parameter No. 1804 is set to 1.

[Example] When the settings indicated below are made, and an unexpected disturbance torque is detected on the 6th axis, the movements on the 2nd axis, 4th axis, 6th axis, and 7th axis are stopped. When an unexpected disturbance torque is detected on the 4th axis, the movements on the 4th axis and the 7th axis are stopped.

Parameter No. 1881	Setting value
(1st axis)	1
(2nd axis)	2
(3rd axis)	1
(4th axis)	0
(5th axis)	3
(6th axis)	2
(7th axis)	0

- Servo axis

	#7	#6	#5	#4	#3	#2	#1	#0
2015			TDOUT					

[Input type] Parameter input
[Data type] Bit axis

#5 TDOUT Select output to the check board (for each axis)
0: Output the torque command to the check board.
1: Output the estimated load torque to the check board.

	#7	#6	#5	#4	#3	#2	#1	#0
2016								ABNT

[Input type] Parameter input
[Data type] Bit axis

#0 ABNT Unexpected disturbance torque detection function (for each axis) is
0: Disabled
1: Enabled
When using the unexpected disturbance torque detection alarm function and estimated disturbance torque output function, enable the unexpected disturbance torque detection function.

2050	Velocity control observer

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 32767
Be sure to set this parameter to 3559.
When using the velocity loop observer (by setting bit 2 (OBEN) of parameter No. 2003 to 1), set 956 in this parameter.

2051	Velocity control observer

[Input type] Parameter input

[Data type] Word axis
[Valid data range] 0 to 32767
Be sure to set this parameter to 3329.
When using the velocity loop observer (by setting bit 2 (OBEN) of parameter No. 2003 to 1), set 510 in this parameter.

2103	Retraction distance upon the detection of an unexpected disturbance torque

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] 0 to 32767
This parameter specifies the distance by which the tool is to be retracted, by reversing the motor, if an unexpected disturbance torque is detected.
When the motor is rotating at low speed, however, the tool may be retracted too far. To prevent this, the motor is stopped, instead of being reversed, while the specified feedrate is less than the value listed in the table below.
When this parameter is set to value A, the detection of an unexpected disturbance torque causes the tool to be retracted in the reverse direction by the distance A, then stopped, if the specified feedrate is equal to or greater than the value listed below.

Detection unit	Feedrate
10 μm	A / 0.8 mm/ min
1 μm	A / 8 mm/ min
0.1 μm	A / 80 mm/ min
0.01 μm	A / 800 mm/ min
0.001 μm	A / 8000 mm/ min

When this parameter is set to 0, the motor stops immediately upon the detection of unexpected disturbance torque.

2104	Threshold for unexpected disturbance torque detection alarm

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Torque command unit
[Valid data range] 0 to 7282 (The maximum motor torque is 7282, regardless of the motor type.)
This parameter specifies the threshold load torque at which an unexpected disturbance torque detection alarm is issued.
Monitor the load torque by setting bit 5 (TDOUT) of parameter No. 2015 to 1 then, for this parameter No. 2104, set a value larger than the maximum monitored torque. An output of 4.4 V is equivalent to 7282 in the units of this parameter.

2142	Threshold for unexpected disturbance torque detection alarm for rapid traverse

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Torque command unit
[Valid data range] 0 to 7282 (The maximum motor torque is 7282, regardless of the motor type.)
This parameter specifies the threshold load torque at which an unexpected disturbance torque detection alarm is issued.
Monitor the load torque by setting bit 5 (TDOUT) of parameter No. 2015 to 1 then, for this parameter No. 2104, set a value larger than the maximum monitored torque. An output of 4.4 V is equivalent to 7282 in the units of this parameter.
This parameter is enabled when bit 3 (ABG0) of parameter No. 2200 = 1.

	#7	#6	#5	#4	#3	#2	#1	#0
2200					ABG0			

[Input type] Parameter input
[Data type] Bit axis

#3 ABG0 The unexpected disturbance torque detection function for cutting or rapid traverse is:
0: Disabled.
1: Enabled.
When this function is disabled, the threshold value during cutting feed is used (parameter No. 2104).

	#7	#6	#5	#4	#3	#2	#1	#0
2215			ABDSW					

[Input type] Parameter input
[Data type] Bit axis

#5 ABDSW The Unexpected disturbance torque detection function for a specified axis:
0: Cannot be disabled.
1: Can be disabled.

- Parameters for spindle

	#7	#6	#5	#4	#3	#2	#1	#0
4015							SPLDMT	

[Input type] Parameter input
[Data type] Bit spindle

#1 SPLDMT Spindle load torque monitor function
0: The spindle load torque monitor function is disabled.
1: The spindle load torque monitor function is enabled.

4247	Magnetic flux compensation time constant for spindle load torque monitor

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] msec
[Valid data range] 0 to 8192

This parameter is used to compensate the delay in the generation of magnetic flux in the spindle motor relative to the specified value. When 0 is set, it is assumed that the generation of magnetic flux is not delayed.
Standard setting depends of the motor model.

4248	Spindle load torque monitor constant

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767

This constant is determined by the maximum output torque and inertia of the motor. It is used for observer processing.
Standard setting depends of the motor model.

4249	Observer gain 1 for spindle load torque monitor

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767
Standard setting is 500.

4250	Observer gain 2 for spindle load torque monitor

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767
Standard setting is 500.

4341	Threshold for unexpected disturbance torque detection alarm

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 0.01%
[Valid data range] 0 to 10000
This parameter specifies the threshold load torque at which an unexpected disturbance torque detection alarm is issued for the spindle. Set a percentage (in units of 0.01 %) for the maximum output torque of the motor. When 0 is set, no Unexpected disturbance torque detection alarm is issued for the spindle.

Alarm and message

- Alarm related to servo axis

Number	Message	Description
SV0409	DETECT ABNORMAL TORQUE	An abnormal load was detected on the servo motor, or during Cs axis or spindle positioning. The alarm can be canceled by RESET.

- Alarm related to spindle

Number	Message	Description
SP0754	ABNORMAL TORQUE	An abnormal load was detected in a spindle motor. The alarm can be canceled by RESET.

Limitation

The following limitations are imposed on the abnormal load detection group function:
<1> Cannot be used with axes on which inter-path synchronous/composite control or inter-path superimposed control is performed.
<2> Cannot be used with axes belonging to different paths.
<3> Cannot be used with rigid tapping with spindles belonging to different paths.
<4> Does not support the inter-path flexible synchronization control function.
<5> Does not support the 3-dimensional rigid tapping function.

2.10 MACHINING CONDITION SELECTION FUNCTION

Overview

For AI contour control function, parameters which place importance on feedrate or precision are set and a precision level is specified according to the machining condition during machining. By using these parameters and level, parameters which satisfy the condition can automatically be calculated for machining.

Format

- Changing the precision level by programming

The precision level can be changed not only by using the precision level selection screen but also by programming the following format:

G05.1 Q1 Rx ;
x : Level (1 to 10)

Also with the format used for, high precision contour control, and AI high precision contour control, the precision level can be changed.

G05 P10000 Rx ;
x : Level (1 to 10)
G08 P1 Rx ;
x : Level (1 to 10)

⚠ **CAUTION**
Once a level is specified, it is kept valid even after the AI contour control mode is canceled.

Default level

By setting the default level for the precision level in parameter No. 11687, a return to the level set in parameter No. 11687 is made during power-off or a reset.
Set the default level to an integer from 1 to 10.
When the default level is set to 0, the current level is maintained.

Parameter

11687	Standard value of precision level when AI contour control is used

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 10

Set standard value of precision level when AI contour control is used.
When the power is turned on or the system is reset, the precision level
0 : keeps its value.
1 to 10 : becomes the level set to this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
13600	MSA							MCR

[Input type] Parameter input
[Data type] Bit path

#0 MCR When an allowable acceleration rate adjustment is made with the machining condition selection function or (machining parameter adjustment screen, precision level selection screen), parameter No. 1735 for the deceleration function based on acceleration in circular interpolation is:
0: Modified.
1: Not modified.

#7 MSA When the machining condition selection function or is used, the acceleration rate change time (bell-shaped) (LV1, LV10) is:

0: Set using parameter Nos. 13612 and 13613.
1: Set using parameter Nos. 13662 and 13663.

	#7	#6	#5	#4	#3	#2	#1	#0
13601								MPR

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 MPR The machining parameter adjustment screen is:
0: Displayed.
1: Not displayed.
Even if 1 is set in this parameter bit, the precision level selection screen for the machining condition selecting function are displayed.

13610	**Acceleration rate for look-ahead acceleration/deceleration before interpolation in AI contour control (precision level 1)**
13611	**Acceleration rate for look-ahead acceleration/deceleration before interpolation in AI contour control (precision level 10)**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)
Each of these parameters sets an acceleration rate for look-ahead acceleration/deceleration before interpolation in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13612	**Acceleration rate change time (bell-shaped) when AI contour control is used (precision level 1)**
13613	**Acceleration rate change time (bell-shaped) when AI contour control is used (precision level 10)**

[Input type] Parameter input
[Data type] Byte path
[Unit of data] msec
[Valid data range] 0 to 127
Each of these parameters sets an acceleration rate change time (bell-shaped) in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13614	**Allowable acceleration rate change amount for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration (precision level 1)**
13615	**Allowable acceleration rate change amount for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration (precision level 10)**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)

Each of these parameters sets an allowable acceleration rate change amount per 1 ms for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration during AI contour control.

Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13616	**Allowable acceleration rate change amount for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration in successive linear interpolation operations (precision level 1)**
13617	**Allowable acceleration rate change amount for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration in successive linear interpolation operations (precision level 10)**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)

Each of these parameters sets an allowable acceleration rate change amount per 1 ms for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration in successive linear interpolation operations during AI contour control.

Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

NOTE

1 For an axis with 0 set in this parameter, parameter No. 13614 and No. 13615 (allowable acceleration rate change amount in speed control based on acceleration rate change under control on the rate of change of acceleration) are valid.

2 For an axis with 0 set in parameter No. 13614 and No. 13615 (allowable acceleration rate change amount in speed control based on acceleration rate change under control on the rate of change of acceleration), speed control based on acceleration rate change is disabled, so that the specification of this parameter has no effect.

13618	**Rate of change time of the rate of change of acceleration in smooth bell-shaped acceleration/deceleration before interpolation when AI contour control is used (precision level 1)**
13619	**Rate of change time of the rate of change of acceleration in smooth bell-shaped acceleration/deceleration before interpolation when AI contour control is used (precision level 10)**

[Input type] Parameter input
[Data type] Byte path

[Unit of data] %

[Valid data range] 0 to 50

Each of these parameters sets the rate (percentage) of the change time of the rate of change of acceleration to the change time of acceleration rate change in look-ahead smooth bell-shaped acceleration/deceleration before interpolation during AI contour control.

Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

NOTE

When 0 or a value not within the valid data range is set in this parameter, look-ahead smooth bell-shaped acceleration/deceleration before interpolation is not performed.

13620	**Allowable acceleration rate when AI contour control is used (precision level 1)**
13621	**Allowable acceleration rate when AI contour control is used (precision level 10)**

[Input type] Parameter input

[Data type] Real axis

[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] Refer to the standard parameter setting table (D)

(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)

Each of these parameters sets an allowable acceleration rate in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13622	**Time constant for acceleration/deceleration after interpolation when AI contour control is used (precision level 1)**
13623	**Time constant for acceleration/deceleration after interpolation when AI contour control is used (precision level 10)**

[Input type] Parameter input

[Data type] Word axis

[Unit of data] msec

[Valid data range] 1 to 512

Each of these parameters sets a time constant for acceleration/ deceleration after interpolation when AI contour control is used. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13624	**Corner speed difference when AI contour control is used (precision level 1)**
13625	**Corner speed difference when AI contour control is used (precision level 10)**

[Input type] Parameter input

[Data type] Real axis

[Unit of data] mm/min, inch/min, deg/min (machine unit)

[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] Refer to the standard parameter setting table (C)

(When the increment system is IS-B, 0.0 to +999000.0)

Each of these parameters sets an allowable speed difference for speed determination based on corner speed difference in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13626	Maximum cutting speed when AI contour control is used (precision level 1)
13627	Maximum cutting speed when AI contour control is used (precision level 10)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Each of these parameters sets a maximum cutting speed in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13628	Parameter number corresponding to arbitrary item 1 when AI contour control is used
13629	Parameter number corresponding to arbitrary item 2 when AI contour control is used

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 65535
These parameters set the parameter numbers corresponding to arbitrary items 1 and 2.

NOTE

1 The parameter numbers corresponding to the following cannot be specified:
   - Bit parameters
   - Spindle parameters Nos. 4000 to 4799
   - Parameters of real number type
   - Parameters that require power-off (for which the alarm PW0000 is issued)
   - Nonexistent parameters

2 When these parameters are set, the power must be turned off before operation is continued.

13630	Value with emphasis on speed (precision level 1) of the parameter corresponding to arbitrary item 1 when AI contour control is used
13631	Value with emphasis on speed (precision level 1) of the parameter corresponding to arbitrary item 2 when AI contour control is used
13632	Value with emphasis on precision (precision level 10) of the parameter corresponding to arbitrary item 1 when AI contour control is used
13633	Value with emphasis on precision (precision level 10) of the parameter corresponding to arbitrary item 2 when AI contour control is used

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Depend on the type of parameter for an arbitrary item
[Valid data range] Depend on the type of parameter for an arbitrary item

Each of these parameters sets a value with emphasis placed on speed or precision for a parameter.

13634	Precision level currently selected when AI contour control is used

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 10
This parameter sets the level currently selected.

13662	Acceleration rate change time (bell-shaped) when AI contour control is used (precision level 1), range extended
13663	Acceleration rate change time (bell-shaped) when AI contour control is used (precision level 10), range extended

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 200
Each of these parameters sets an acceleration rate change time (bell-shaped) in AI contour control. Set a value (precision 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

2.11 MALFUNCTION PREVENT FUNCTIONS

Overview

These functions monitor the CNC internal status and check that related data is within the allowable range. If an invalid state due to a deteriorated hardware component or noise is detected, these functions stop the machine with an alarm to prevent any malfunction.

Explanation

Each function is detailed below.

- Checking the maximum speed of the servo motor

This function checks whether the maximum speed specified in parameter No. 12255 is exceeded. If the maximum speed is exceeded, this function issues alarms DS0004 and PW0006 and stops the machine. When the parameter is set to 0, this function checks whether the maximum speed (999000 for IS-B) is exceeded.

- Checking the maximum acceleration of the servo motor

This function checks whether the maximum acceleration specified in parameter No. 12256 is exceeded. If the maximum acceleration is exceeded, this function issues alarms DS0005 and PW0006 and stops the machine. When the parameter is set to 0, this function does not check the maximum acceleration.

- Checking the maximum speed of the spindle motor

This function checks whether the maximum speed of the spindle motor is exceeded. If the maximum speed is exceeded, this function issues alarms IE0007 and PW0006 and outputs S0. The parameter which specifies the maximum speed of each spindle is used.

- Checking the stored stroke limit at the end point

This function makes a stored stroke check immediately before pulses are output in addition to a normal stored stroke check. If the stored stroke limit is exceeded, this function issues one of alarms IE0001 to

IE0006 and alarm PW0006 and stops the machine. The parameters for normal stored stroke checks are used.

- Monitoring execution of NC command analysis

This function monitors NC command analysis to check the execution counter and reference area for validity. If invalid data is detected, this function issues one of alarms DS0006 to DS0013 and alarm PW0006 and stops the machine.

- Monitoring execution of acceleration/deceleration after interpolation

This function monitors the execution counter during execution of linear/bell-shaped acceleration/deceleration after interpolation to check the counter for validity. If invalid data is detected, this function issues alarms IE0008 and PW0006 and stops the machine.

Parameter

12255	Maximum servo motor speed

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets a maximum servo motor speed. When the value set in this parameter is exceeded, the servo motor stops with the alarm DS0004. When 0 is set in this parameter, the specification of a maximum allowable value (999000 for IS-B) is assumed.

12256	Maximum servo motor acceleration rate

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)

This parameter sets a maximum servo motor acceleration rate. When the value set in this parameter is exceeded, the servo motor stops with the alarm DS0005. When 0 is set in this parameter, alarm check is not performed.

Alarm and message

The machine is decelerated and stopped when a DS alarm is issued or is immediately stopped when an IE alarm is issued to prevent any malfunction.
Check the block in which the machine is stopped. If you cannot find the cause, contact FANUC without releasing the alarm state.

Number	Message	Description
DS0004	EXCESS MAXIMUM FEEDRATE	The malfunction prevention function detected the command in which a value exceeding the maximum speed was specified.
DS0005	EXCESS MAXIMUM ACCELERATION	The malfunction prevention function detected the command in which a value exceeding the maximum acceleration was specified.
DS0006	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0007	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.

Number	Message	Description
DS0008	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0009	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0010	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an illegal reference area.
DS0011	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an illegal reference area.
DS0012	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an illegal reference area.
DS0013	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an illegal reference area.
IE0001	+ OVERTRAVEL (SOFT 1)	The malfunction prevention function detected that stored stroke check 1 on the positive side was exceeded.
IE0002	- OVERTRAVEL (SOFT 1)	The malfunction prevention function detected that stored stroke check 1 on the negative side was exceeded.
IE0003	+ OVERTRAVEL (SOFT 2)	The malfunction prevention function detected that stored stroke check 2 on the positive side was exceeded.
IE0004	- OVERTRAVEL (SOFT 2)	The malfunction prevention function detected that stored stroke check 2 on the negative side was exceeded.
IE0005	+ OVERTRAVEL (SOFT 3)	The malfunction prevention function detected that stored stroke check 3 on the positive side was exceeded.
IE0006	- OVERTRAVEL (SOFT 3)	The malfunction prevention function detected that stored stroke check 3 on the negative side was exceeded.
IE0007	EXCESS MAXIMUM REV. DATA	The malfunction prevention function detected the command in which a value exceeding the maximum speed was specified.
IE0008	ILLEGAL ACC/DEC	The malfunction prevention function detected the acceleration/deceleration error.

2.12 OPERATOR ERROR PREVENT FUNCTIONS

Overview

If an invalid value is set for tool offset or the machine is operated improperly, a workpiece may be cut incorrectly or a tool may be damaged. If data is lost due to an operator error, time is required for recovery. The operator error prevent functions include the following functions to prevent an unintentional operator operation (called an operator error below).

1. Functions operating during data setting
   - Checking whether offset data is within the valid setting range
   - Reconfirming incremental input operation
   - Disabling absolute input using soft keys to prevent an operator error related to absolute and incremental input
   - Reconfirming deletion of a program or all data
   - Reconfirming data update during setting
2. Functions operating during execution
   - Highlighting updated modal data
   - Displaying the status of the execution block before execution of a program
   - Displaying the axis status such as the mirror image enable state or interlock enable state
   - Checking the start of a program from a midpoint
   - Checking whether offset data is within the valid setting range
   - Checking the maximum incremental amount

Explanation

- Functions operating during data setting

The following functions are provided to prevent an operation error during data setting.

- Checking of the input data range
- Reconfirming of incremental input
- Prohibiting of soft key absolute input
- Reconfirming of program deletion
- Reconfirming of all data deletion
- Reconfirming of data update during setting

For details on these functions, refer to the OPERATOR’S MANUAL.

- Functions related to checking before execution

The following functions are provided to prevent an operation error during execution.

- Update display of modal information
- Start check signal
- Status display of axes
- Reconfirming of midway block start
- Checking of the execution data range
- Checking of the maximum incremental amount

For details of functions other than the start check signal and bit 0 of diagnosis information No. 3570, refer to the OPERATOR’S MANUAL.

- Start check signal

If cycle start is performed when the start check signal STCHK <Gn408.0> is 1, the stop state is entered with the remaining traverse distance and modal of the execution block indicated. Performing cycle start again starts the execution.
An operation error is prevented because the status of the execution block can be checked before execution.
Using the start check signal together with the update display of modal information shown above makes checking easier.

- During a memory operation stop

If memory operation is stopped due to the reconfirming of midway block start, bit 0 of diagnosis information No. 3570 is set to 1, so that the state can be confirmed.
In a multi-path system, bit 0 of diagnosis information No. 3570 is set to 1 on only the path on the cursor is position in the middle of the program.

Signal

Start check signal STCHK<Gn408.0>

[Classification] Input signal

[Function] The traverse distance and modal can be checked before execution.

[Operation] When this signal is “1”, the remaining traverse distance and modal is displayed before executing a block and then execution stops. Performing cycle start again starts the execution.

Middle block start signal MBSO<Fn534.4>

[Classification] Output signal

[Function] Indicates that memory operation is stopped by the reconfirming of middle block start of the operator error prevent function.

[Output cond.] The signal is set to “1” when:

- Memory operation is stopped by the reconfirming of middle block start of Operator error prevent function.

In a multi-path system, the bit is set to 1 on only the path on which the cursor is position in the middle of the program.

The signal is set to “0” when:

- Memory operation is not stopped by the reconfirming of middle block start of Operator error prevent function.

When bit 0 (MBO) of parameter No.10336 is set to 1, this signal is enabled.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn408								STCHK
Fn534				MBSO				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3109						IKY		

[Input type] Parameter input
[Data type] Bit path

#2 IKY On the tool offset screen and workpiece shift screen (T series), soft key [INPUT] is:
0: Displayed.
1: Not displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3191					SSF	WSI		

[Input type] Parameter input
[Data type] Bit path

#2 WSI On the workpiece zero point offset screen, the soft key [INPUT] is:
0: Displayed.
1: Not displayed.

#3 SSF On the setting screen, the soft key for confirming data input is:
0: Not displayed.
1: Displayed.

10000	Lower limit 1 of tool offsets No. 01
to	to
10019	Lower limit 1 of tool offsets No. 20

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

These parameters set the lower limits of the following offset values:

- T series, without tool geometry/wear offsets, X-axis offset
- T series, with tool geometry/wear offsets, X-axis and geometry offsets
- M series, tool offset memory A offset
- M series, tool offset memory C, geometry, and length offsets

10020	**Upper limit 1 of tool offsets No. 01**
to	**to**
10039	**Upper limit 1 of tool offsets No. 20**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the upper limits of the following offset values:
- T series, without tool geometry/wear offsets, X-axis offset
- T series, with tool geometry/wear offsets, X-axis and geometry offsets
- M series, tool offset memory A offset
- M series, tool offset memory C, geometry, and length offsets

10040	**Lower limit 2 of tool offsets No. 01**
to	**to**
10059	**Lower limit 2of tool offsets No. 20**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the lower limits of the following offset values:
- T series, without tool geometry/wear offsets, Z-axis offset
- T series, with tool geometry/wear offsets, Z-axis and geometry offsets
- M series, tool offset memory C, geometry, and radius offsets

10060	**Upper limit 2 of tool offsets No. 01**
to	**to**
10079	**Upper limit 2 of tool offsets No. 20**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the upper limits of the following offset values:
- T series, without tool geometry/wear offsets, Z-axis offset
- T series, with tool geometry/wear offsets, Z-axis and geometry offsets
- M series, tool offset memory C, geometry, and radius offsets

10080	**Lower limit 3 of tool offsets No. 01**
to	**to**
10099	**Lower limit 3 of tool offsets No. 20**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the lower limits of the following offset values:

- T series, without tool geometry/wear offsets, tool nose radius offset
- T series, with tool geometry/wear offsets, tool nose radius and geometry offsets

10100	**Upper limit 3 of tool offsets No. 01**
to	**to**
10119	**Upper limit 3 of tool offsets No. 20**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the upper limits of the following offset values:

- T series, without tool geometry/wear offsets, tool nose radius offset
- T series, with tool geometry/wear offsets, tool nose radius and geometry offsets

10120	**Lower limit 4 of tool offsets No. 01**
to	**to**
10139	**Lower limit 4 of tool offsets No. 20**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the lower limits of the following offset values:

- T series, with tool geometry/wear offsets, X-axis and wear offsets
- M series, tool offset memory C, wear, and length offsets

10140	**Upper limit 4 of tool offsets No. 01**
to	**to**
10159	**Upper limit 4 of tool offsets No. 20**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the upper limits of the following offset values:

- T series, with tool geometry/wear offsets, X-axis and wear offsets
- M series, tool offset memory C, wear, and length offsets

10160	**Lower limit 5 of tool offsets No. 01**
to	**to**
10179	**Lower limit 5 of tool offsets No. 20**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)

[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the lower limits of the following offset values:
- T series, with tool geometry/wear offsets, Z-axis and wear offsets
- M series, tool offset memory C, wear, and radius offsets

10180	**Upper limit 5 of tool offsets No. 01**
to	**to**
10199	**Upper limit 5 of tool offsets No. 20**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the upper limits of the following offset values:
- T series, with tool geometry/wear offsets, Z-axis and wear offsets
- M series, tool offset memory C, wear, and radius offsets

10200	**Lower limit 6 of tool offsets No. 01**
to	**to**
10219	**Lower limit 6 of tool offsets No. 20**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the lower limits of the following offset values:
- T series, with tool geometry/wear offsets, tool noise radius and wear offsets

10220	**Upper limit 6 of tool offsets No. 01**
to	**to**
10239	**Upper limit 6 of tool offsets No. 20**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the upper limits of the following offset values:
- T series, with tool geometry/wear offsets, tool noise radius and wear offsets

10240	**Lower limit 1 of a tool offset number range No. 01**
to	**to**
10259	**Lower limit 1 of a tool offset number range No. 20**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to maximum number of offset sets
Each of these parameters sets the lower limit of a tool offset number range.

These parameters correspond to the tool offset lower/upper limits set in parameters Nos. 10000 to 10239.

10260	**Upper limit 1 of a tool offset number range No. 01**
to	**to**
10279	**Upper limit 1 of a tool offset number range No. 20**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to maximum number of offset sets

Each of these parameters sets the upper limit of a tool offset number range.
These parameters correspond to the tool offset lower/upper limits set in parameters Nos. 10000 to 10239.

10280	**Lower limit 7 of tool offsets No. 01**
to	**to**
10283	**Lower limit 7 of tool offsets No. 04**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

These parameters set the lower limits of the following offset values:

- T series, without tool geometry/wear offsets, Y-axis offset
- T series, with tool geometry/wear offsets, Y-axis and geometry offsets

10284	**Upper limit 7 of tool offsets No. 01**
to	**to**
10287	**Upper limit 7 of tool offsets No. 04**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

These parameters set the upper limits of the following offset values:

- T series, without tool geometry/wear offsets, Y-axis offset
- T series, with tool geometry/wear offsets, Y-axis and geometry offsets

10288	**Lower limit 8 of tool offsets No. 01**
to	**to**
10291	**Lower limit 8 of tool offsets No. 04**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

These parameters set the lower limits of the following offset values:

- T series, with tool geometry/wear offsets, Y-axis and wear offsets

10292	**Upper limit 8 of tool offsets No. 01**
to	**to**
10295	**Upper limit 8 of tool offsets No. 04**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

These parameters set the upper limits of the following offset values:

- T series, with tool geometry/wear offsets, Y-axis and wear offsets

10296	**Lower limit 2 of a tool offset number range No. 01**
to	**to**
10299	**Lower limit 2 of a tool offset number range No. 04**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to maximum number of offset sets

Each of these parameters sets the lower limit of a tool offset number range.
These parameters correspond to the tool offset lower/upper limits set in parameters Nos. 10280 to 10295.

10300	**Upper limit 2 of a tool offset number range No. 01**
to	**to**
10303	**Upper limit 2 of a tool offset number range No. 04**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to maximum number of offset sets

Each of these parameters sets the upper limit of a tool offset number range.
These parameters correspond to the tool offset lower/upper limits set in parameters Nos. 10280 to 10295.

10304	**Lower limit of workpiece zero point offsets No. 01**
to	**to**
10309	**Lower limit of workpiece zero point offsets No. 06**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Each of these parameters sets the lower limit of workpiece zero point offset values.

10310	**Upper limit of workpiece zero point offsets No. 01**
to	**to**
10315	**Upper limit of workpiece zero point offsets No. 06**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Each of these parameters sets the upper limit of workpiece zero point offset values.

10316	**Lower limit of a workpiece zero point offset range No. 01**
to	**to**
10321	**Lower limit of a workpiece zero point offset range No. 06**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to maximum number of offset sets
Each of these parameters sets the lower limit of a workpiece zero point offset range. For an additional workpiece coordinate system, set a value after adding 1000.
These parameters correspond to the workpiece zero point offset lower/upper limits set in parameters Nos. 10304 to 10315.

10322	**Upper limit of a workpiece zero point offset range No. 01**
to	**to**
10327	**Upper limit of a workpiece zero point offset range No. 06**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to maximum number of offset sets
Each of these parameters sets the upper limit of a workpiece zero point offset range. For an additional workpiece coordinate system, set a value after adding 1000.
These parameters correspond to the workpiece zero point offset lower/upper limits set in parameters Nos. 10304 to 10315.

10328	**Lower limit of workpiece shifts**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets a workpiece shift lower limit.

10329	**Upper limit of workpiece shifts**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets a workpiece shift upper limit.

	#7	#6	#5	#4	#3	#2	#1	#0
10330		**ASD**		**MID**	**HSC**	**ADC**	**PDC**	**IIC**

[Input type] Parameter input
[Data type] Bit

#0 IIC At the time of incremental input, a confirmation message is:
0: Displayed.
1: Not displayed.

#1 PDC At the time of program deletion, a confirmation message is:
0: Displayed.
1: Not displayed.

#2 ADC At the time of deletion of all data, a confirmation message is:
0: Displayed.
1: Not displayed.

#3 HSC When a cycle start is executed halfway in the program, a confirmation message is:
0: Displayed.
1: Not displayed.

#4 MID Updated modal information is:
0: Highlighted.
1: Not highlighted.

#6 ASD Axis state display is:
0: Enabled.
1: Disabled.

10331	Lower limit of external workpiece zero point offsets

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the lower limit of external workpiece zero point offsets.

10332	Upper limit of external workpiece zero point offsets

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the upper limit of external workpiece zero point offsets.

	#7	#6	#5	#4	#3	#2	#1	#0
10335								MSC

[Input type] Parameter input
[Data type] Bit path

#0 MSC The reconfirming of midway block start of operator error prevent function is:
0: Enabled independently for each path.
1: Enabled for the local path and those paths for this parameter is set to 1.

- Additional information

Parameters for input data range checks are assigned as follows:

- Tool offset
  - <1> No tool geometry/wear offset for a lathe system
    Nos. 10000 to 10119, Nos. 10240 to 10279
  - <2> With tool geometry/wear offset for a lathe system
    Nos. 10000 to 10279
  - <3> Tool offset memory A for a machining center system
    Nos. 10000 to 10039, Nos. 10240 to 10279
  - <4> Tool offset memory C (geometry/wear, length/radius) for a machining center system
    Nos. 10000 to 10079, Nos. 10120 to 10199, Nos. 10240 to 10279
- Y-axis tool offset
  Nos. 10280 to 10303
- Workpiece origin offset
  Nos. 10304 to 10327
- Workpiece shift
  Nos. 10328 and 10329

	#7	#6	#5	#4	#3	#2	#1	#0
10336								MBO

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 MBO Middle block start signal MBSO<Fn534.4> is:
0: Disabled.
1: Enabled.

Alarm and message

Number	Message	Description
PS0334	OFFSET IS OUT OF EFFECTIVE RANGE	An offset data which was out of the effective range was specified. (operator error prevention function)
PS0337	EXCESS MAXIMUM INCREMENTAL VALUE	The command value exceeded the maximum amount of incremental. (operator error prevention function)

Diagnosis display

	#7	#6	#5	#4	#3	#2	#1	#0
3570								MSC

#0 MSC Memory operation is stopped due to the reconfirming of midway block start.
In a multi-path system, the bit is set to 1 on only the path on which the cursor is position in the middle of the program.

Caution

> ⚠ **CAUTION**
> The input data range check function does not check the values related to the second geometry offset function. The second geometry offset function, which is used to set values specific to the machine, is not assumed to be modified by end users.

2.13 Improvement of stop distance reduction function at power failure

Overview

Improvement of stop distance reduction function at power failure becomes effective by setting bit 1 (PFSx) of parameter No.11820 to 1. When the power supply detects a power failure, the axis immediately stops (alarm SV1094) by this function.
Stop method of the axis can be changed to deceleration stop (alarm DS0099) by setting bit 2 (PFD) of parameter No.11808 and bit 1 (PFSx) of parameter No.11820 to 1.
To stop the axis at power failure, conventional stop distance reduction function at power failure requires the PMC ladder or the wiring.
This function makes the PMC ladder and the wiring unnecessary.

Explanation

If the power failure is detected by the power supply, the power failure detection signals XPFL1 – XPFL8 <Fn374> is set to “0”.
When the power failure detection signals XPFL1 – XPFL8 is “0”, the axis immediately stops (alarm SV1094) by setting bit 1 (PFSx) of parameter No.11820 to 1
Stop method of the axis can be changed to deceleration stop (alarm DS0099) by setting bit 2 (PFD) of parameter No.11808 and bit 1 (PFSx) of parameter No.11820 to 1.

Table 2.13 (a) Stop method of improvement of stop distance reduction function at power failure

Bit 1 (PFSx) of parameter No.11820	Bit 2 (PFD) of parameter No.11808	Stop method
0	0	This function is invalid
0	1	This function is invalid
1	0	Immediately stop (alarm SV1094)
1	1	Deceleration stop (alarm DS0099)

As for this function, immediately stop or deceleration stop is executed by the power failure detection signals from the power supply. Therefore, the PMC ladder and the wiring become unnecessary by this function.

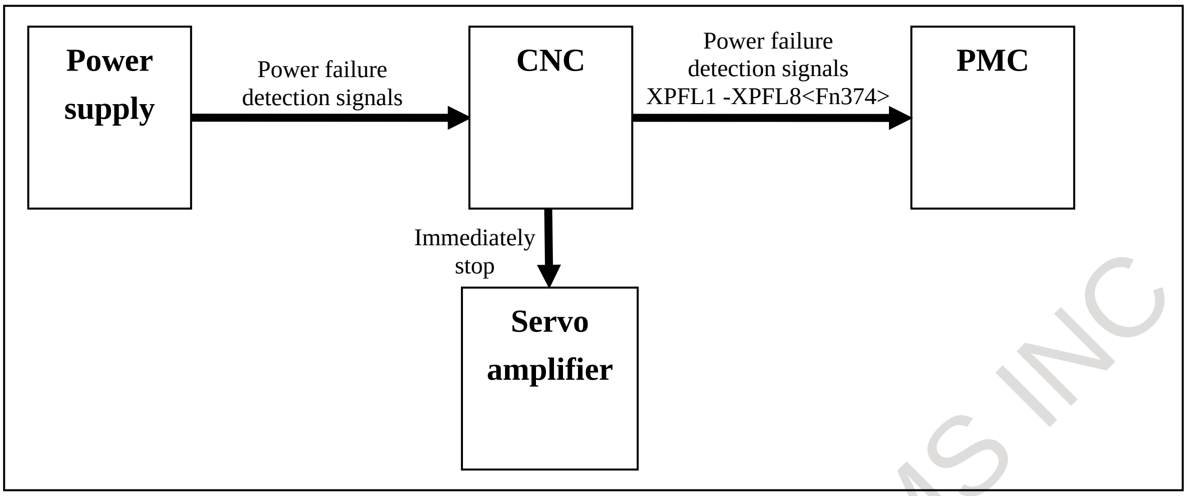


Fig. 2.13 (a) Immediately stops by this function (alarm SV1094)

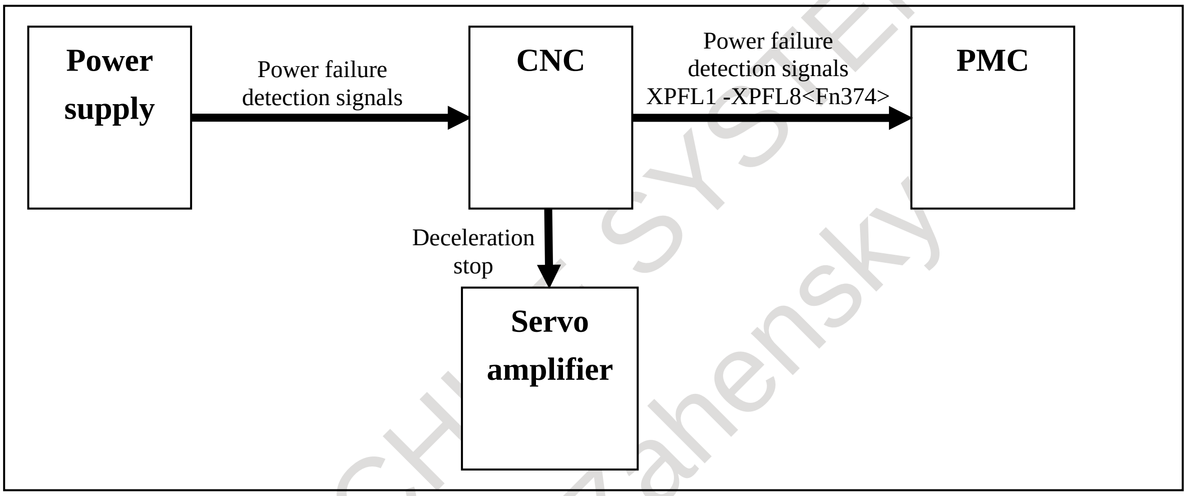


Fig. 2.13 (b) Deceleration stops by this function (alarm DS0099)

To stop the axis at power failure, conventional stop distance reduction function at power failure required like following 1 – 3 the PMC ladder or the wiring.

1 If immediately stops of the axis are executed by emergency stop, it was necessary to make the PMC ladder. The PMC ladder executes the emergency stop by making emergency stop *ESP <Gn008.4> to 0 when the power failure detection signals XPFL1 – XPFL8 <Fn374> is 0.

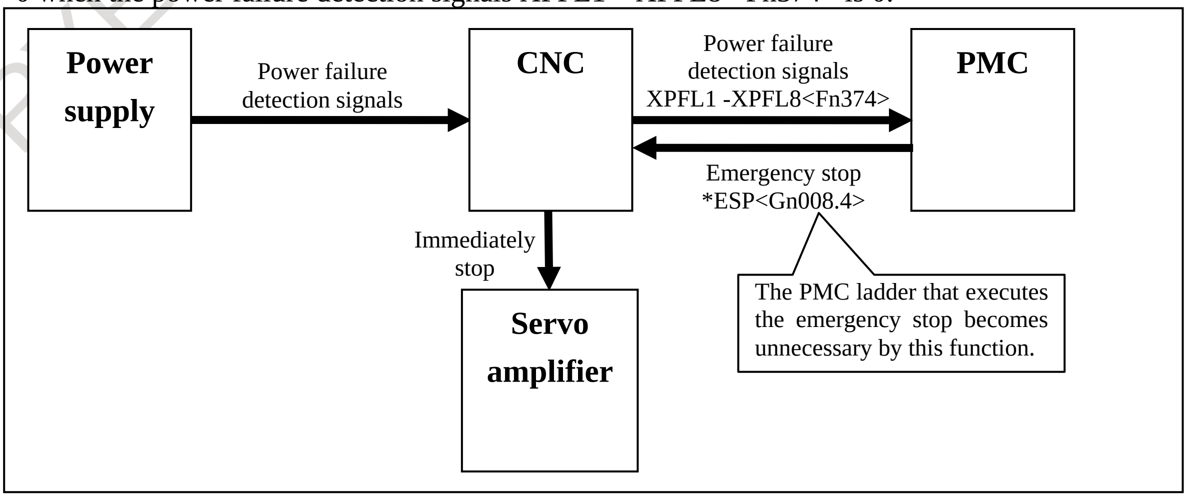


Fig. 2.13 (c) Immediately stops of the axis by emergency stop (PMC ladder)

2 If deceleration stops of the axis are executed by feed hold signal, it was necessary to make the PMC ladder. The PMC ladder executes the feed hold stop by making feed hold signal *SP <Gn008.5> to 0 when the power failure detection signals XPFL1 – XPFL8 <Fn374> is 0.

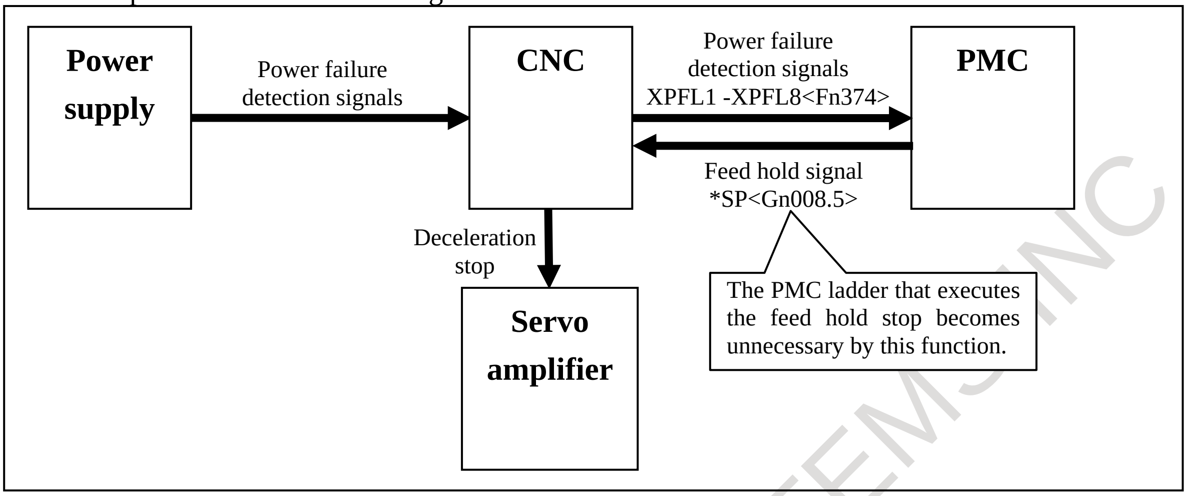


Fig. 2.13 (d) Deceleration stops of the axis by feed hold signal (PMC ladder)

3 If immediately stops of the axis are executed by emergency stop that uses power failure detection signal (contact signal) from the power supply, it was necessary to make the wiring. The wiring wires emergency stop *ESP <X8.4>,<X8.0>,<X8.1> of PMC for the power failure detection signal (contact signal) from the power supply.
Wiring effort can be reduced by using this function though immediately stops of the axis by the following wiring can be achieve the stop at power failure earlier than stop time that used this function.

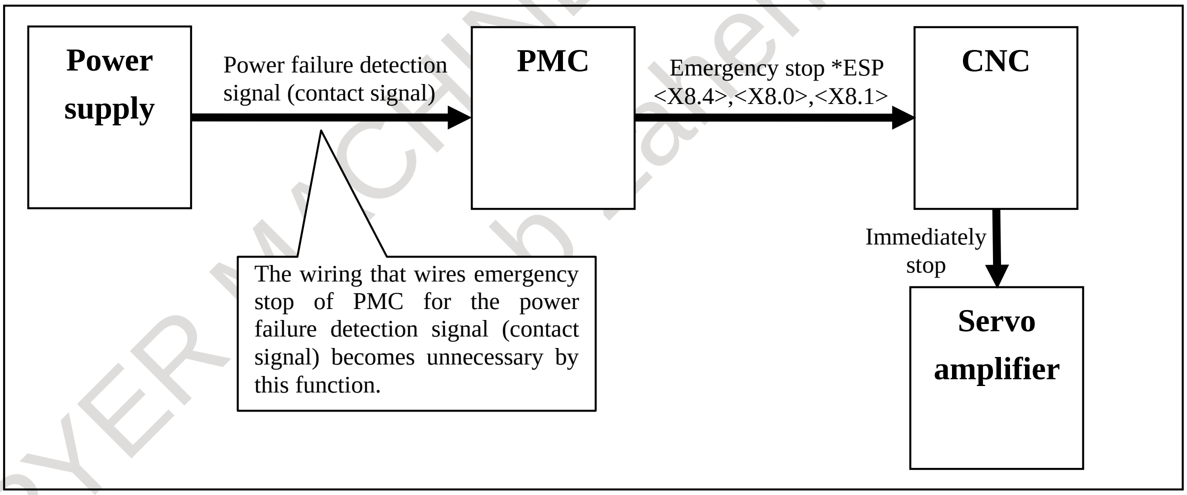


Fig. 2.13 (e) Immediately stops of the axis by emergency stop (Wiring for contact signal)

NOTE
- Improvement of stop distance reduction function at power failure is valid after the servo ready signal SA <Fn000.6> is set to “1” for the first time after power on.
- It enters the servo off state at the same time as generating the alarm at the immediately stops (alarm SV1094), and it shifts to deceleration stop operation with the dynamic brake. In this case, the stopping distance can be shortened by using brake control function (bit 6 (BRKCx) of parameter No.2005) and quick stop at emergency stop (bit 0 (DBSTx) of parameter No.2017). Refer to Servo motor parameter manual (B-65270EN) for details of brake control function and quick stop at emergency stop.
- This function executes the axis stop directly according to the power failure detection signal from power supply without using PMC. Therefore, timing in which the axis stop begins is different from conventional stop distance reduction function at power failure. Use conventional stop distance reduction function at power failure if the axis should stop after operating the machine side according to the power failure detection signals XPFL1 – XPFL8 <Fn374>.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
2005		BRKCx						

[Input type] Parameter input
[Data type] Bit axis

#6 BRKCx Brake control function is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
2017								DBSTx

[Input type] Parameter input
[Data type] Bit axis

#0 DBSTx Specifies whether to enable quick stop type 1 at emergency stop as follows:
0: To disable.
1: To enable.
When using the quick stop at emergency stop function, make the brake control setting for all the axes to which thus function is applied.

NOTE
When only the brake control function is set, a gradual stop occurs with the torque limit set by the torque limit setting function during brake control.
When the quick stop at emergency stop is enabled, a gradual stop occurs with the torque limit set to 100%, so that the stop distance is reduced.

	#7	#6	#5	#4	#3	#2	#1	#0
11808						PFD		

[Input type] Parameter input
[Data type] Bit path

#2 PFD Improvement of stop distance reduction function at power failure is:
0: Immediately stop (alarm SV1094).
1: Deceleration stop (alarm DS0099).

	#7	#6	#5	#4	#3	#2	#1	#0
11820							PFSx	

[Input type] Parameter input
[Data type] Bit axis

#1 PFSx Improvement of stop distance reduction function at power failure is:
0: Disabled.
1: Enabled.

Signal

Emergency stop (input)

*ESP<X008.4> (for the first machine group), <X008.0> (for the second machine group), <X008.1> (for the third machine group), <Gn008.4>

[Classification] Input signal
[Function] Activating an emergency stop signal stops the machine instantly.
[Operation] When the emergency stop signal *ESP turns to "0", the emergency stop is applied to the machine and the CNC is reset. This signal is controlled by the B contacts of a pushbutton switch. The emergency stop signal turns the servo ready signal SA to "0".

Feed hold signal *SP<Gn008.5>

[Classification] Input signal
[Function] Halts automatic operation.
[Operation] When signal *SP is set to 0 during automatic operation, the CNC enters the feed hold state and stops operation. Automatic operation cannot be started when signal *SP is set to 0.
MEM, RMT, or MDI mode

Servo Ready Signal SA<Fn000.6>

[Classification] Output signal
[Function] Signal SA turns to "1" when the servo system is ready to operate. For an axis that is to be braked, release the brake when this signal is "1" and apply the brake when this signal is "0".

Power Failure Detection Signals XPFL1 to XPFL8<Fn374>

[Classification] Output signal
[Function] This signal indicates that power failure is detected by the power supply.
[Operation] This signal turns to 1 at:
- Normal

The signal turns to 0 at:
- Power failure status

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X008				*ESP			(*ESP)	(*ESP)
Gn008			*SP	*ESP				
Fn000		SA						
Fn374	XPFL8	XPFL7	XPFL6	XPFL5	XPFL4	XPFL3	XPFL2	XPFL1

Alarm and message

Number	Message	Description
SV1094	STOP DISTANCE REDUCTION ENABLE	The axis stop is executed by stop distance reduction function at power failure.
DS0099	STOP DISTANCE REDUCTION ENABLE	The axis stop is executed by stop distance reduction function at power failure.

3 MANUAL OPERATION

3.1 JOG FEED/INCREMENTAL FEED

Overview

- Jog feed

In jog mode, setting a feed axis and direction selection signal to “1” moves the tool along the selected axis in the selected direction continuously.
Manual operation is allowed one axis at a time. Up to 3 axes can be selected at a time by setting bit 0 (JAX) of parameter No.1002.
By using bit 6 of parameter No. 7001, the number of axes that can be selected can be increased up to the maximum number of simultaneously controlled axes.

- Incremental feed

In incremental feed mode, setting a feed axis and direction selection signal to “1” moves the tool one step along the selected axis in the selected direction. The minimum distance the tool is moved, is the least input increment. The step can be 10, 100, or 1000 times the least input increment.
Bit 2 (HNT) of parameter No. 7103 can be used to multiply the obtained step by 10.

- Feedrate

The jog feedrate is specified in parameter No.1423.
The jog feedrate can be adjusted with the jog feedrate override dial.
With manual rapid traverse selection signal, the tool can be moved at the rapid traverse rate regardless of the jog feedrate override signal.

Explanation

- Manual per revolution feed

The manual per revolution feed is enabled for jog feed and incremental feed by setting bit 4 (JRV) of parameter No.1402 to 1. During the manual per revolution feed, jog feed and incremental feed is performed at the feedrate that is obtained by multiplying the spindle speed and the jog feedrate override value by the feed amount per revolution specified with the parameter No. 1423. In the case of linear axis (metric input), during manual per revolution feed, the tool is jogged at the following feedrate:
Parameter setting value (mm/rev) × JOG feedrate override × actual spindle speed (rev/min).

NOTE
1 Manual per revolution feed is not available at the setting of without the position coder (setting bit 0 (NPC) of parameter No.1402 to 1 or bit 1 (FPR) of parameter No.3729 to 1).
2 When multi spindle control is enabled and the multiple position coders are used, the target feedback pulses from the position coder for manual per revolution feed is selected with a position coder select signal (PC2SLC<Gn028.7>, PC3SLC<Gn026.0>, PC4SLC <Gn026.1>). And in addition, when multi path control is enabled, an arbitrary position coder can be selected by combining a path spindle feedback selection signals (SLPCA<Gn064.2>, SLPCB<Gn064.3>, SLPCC<Gn403.4>, SLPCD<Gn403.5>). For the selection of a position coder, see the section of "Multi spindle" and "Path spindle control".

Signal

The following signals determine that way the jog feed or incremental feed is executed.

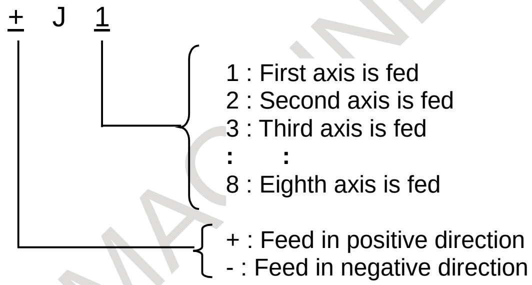
Selection	Jog feed	Incremental feed
Mode selection	MD1, MD2, MD4, MJ	MD1, MD2, MD4, MINC
Selection of the axis to move	+J1, -J1, +J2, -J2, +J3, -J3, ...	
Selection of the direction to move the axis		
Selection of the move amount		MP1, MP2
Selection of feedrate	*JV0 to *JV15, RT, ROV1, ROV2	

The only difference between jog feed and incremental feed is the method of selecting the feed distance. In jog feed, the tool continues to be fed while feed axis and direction selection signals are 1: +J1, -J1, +J2, -J2, +J3, -J3, etc. In incremental feed, the tool is fed by one step.
The distance of the step is selected by the manual handle feed move distance selection signals MP1 and MP2.
For the signals selecting the mode, see "Mode Selection Signals." In selection of the move amount, for manual handle feed amount selection signals MP1 and MP2, see "MANUAL HANDLE FEED". In selection of feedrate, for rapid traverse override signals ROV1 and ROV2, see “Feedrate Override Signals.”
Other signals are described below.

Feed Axis and Direction Selection Signal +J1 to +J8<Gn100>-J1 to -J8<Gn102>

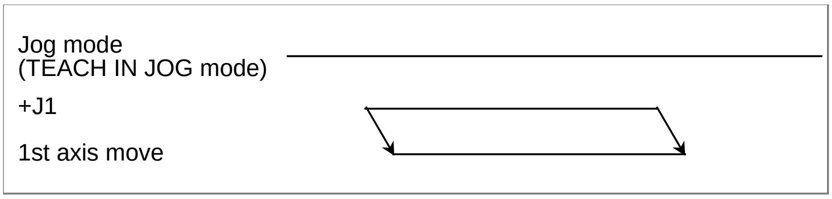
[Classification] Input signal

[Function] Selects a desired feed axis and direction in jog feed or incremental feed. The sign (+ or -) in the signal name indicates the feed direction. The number following J indicates the number of the control axis.



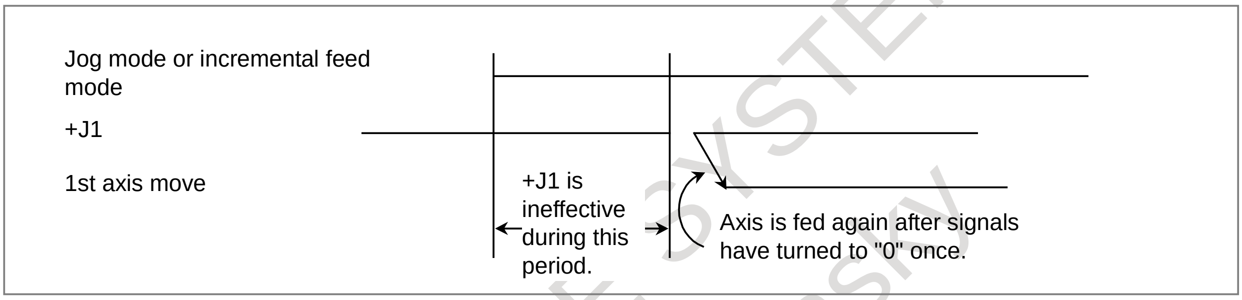
[Operation] When the jog bit is “1”, the control unit operates as described below.

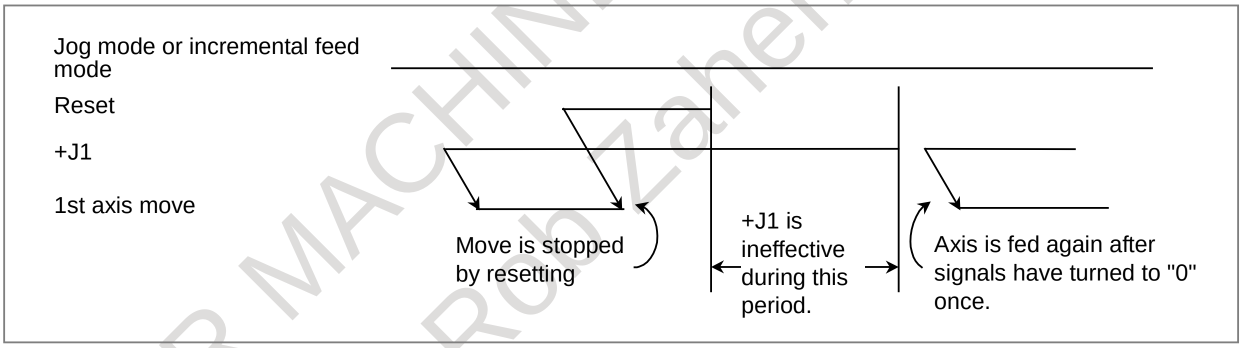
- When jog feed or incremental feed is allowed, the control unit moves the specified axis in the specified direction. In jog feed, the control unit continues to feed the axis while the bit is “1”.



- In incremental feed, the control unit feeds the requested axis by the step distance which is specified by the manual handle feed move distance selection signals MP1, MP2, then the axis stops. Even if the signal is set to “0” while the axis is being fed, the control unit does not stop moving.
  To feed the axis again, set the signal to “0”, then to “1” again.

> ⚠ **CAUTION**
>
> 1 If both the positive direction and negative direction signals of the same axis are simultaneously set to “1”, neither the positive direction nor the negative direction is selected. The control unit assumes that both these signals are set to “0”.
>
> 2 If the feed axis and direction selection signals are set to “1” before the jog feed mode or incremental feed mode is selected, these signals are invalidated. After the jog feed mode or incremental feed mode is selected, set these signal to “0”, then set them to “1” again.
>
> 3 If the control unit is reset while the feed axis and direction selection signals are set to “1” or if a feed axis and direction signal turns to “1” while the control unit is in the reset state, the signal will be ignored even after releasing reset. After the reset state is released, set these signals to “0”, then set them to “1” again.





Manual feedrate override signal *JV0 to *JV15<Gn010,Gn011>

[Classification] Input signal

[Function] Selects a feedrate in jog feed or incremental feed. These signals are in sixteen bit binary code, which corresponds to the override values as follows:

$$\text{Override value (\%)} = 0.01\% \times \sum_{i=0}^{15} \left| 2^i \times Vi \right|$$

where Vi=0 when *JVi is “1”.
Vi=1 when *JVi is “0”.

The override value is assumed to be zero when all of the signals, (*JV0 to *JV15) are set to “1” or “0”. When this occurs, the feed is stopped. The override value can be specified in the range of 0% to 655.34% in units of 0.01%. Some examples are listed below.

*JV0 to *JV15				Override value
12	8	4	0	
1 1 1 1	1 1 1 1	1 1 1 1	1 1 1 1	0
1 1 1 1	1 1 1 1	1 1 1 1	1 1 1 0	0.01
1 1 1 1	1 1 1 1	1 1 1 1	0 1 0 1	0.10
1 1 1 1	1 1 1 1	1 0 0 1	1 0 1 1	1.00
1 1 1 1	1 1 0 0	0 0 0 1	0 1 1 1	10.00
1 1 0 1	1 0 0 0	1 1 1 0	1 1 1 1	100.00
0 1 1 0	0 0 1 1	1 0 1 1	1 1 1 1	400.00
0 0 0 0	0 0 0 0	0 0 0 0	0 0 0 1	655.34
0 0 0 0	0 0 0 0	0 0 0 0	0 0 0 0	0

[Operation] If rapid traverse selection signal RT is “0” during jog feed or incremental feed, actual feedrate is manual feedrate set by parameter No.1423 multiplied by override value selected by this signals.

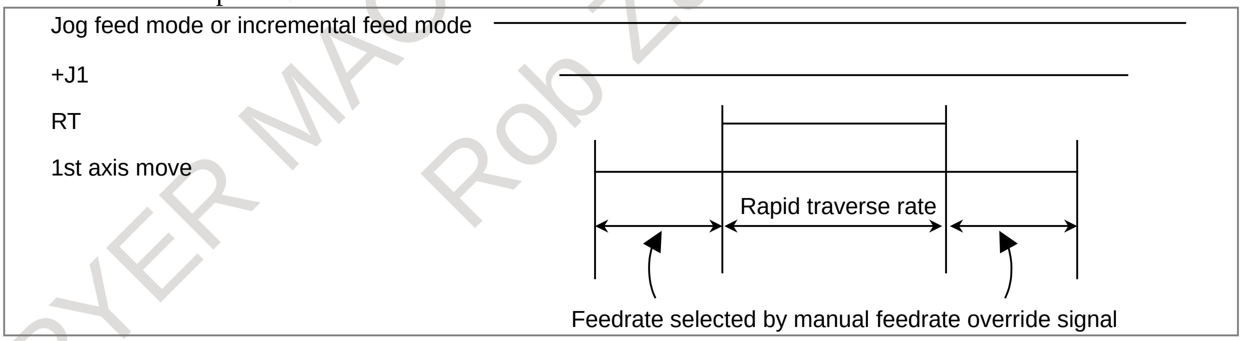
NOTE
The *JVi signals also serve as the override signals during dry run in automatic operation mode.

Manual rapid traverse selection signal RT<Gn019.7>

[Classification] Input signal

[Function] Selects a rapid traverse rate for jog feed or incremental feed.

[Operation] When the signal turns to “1”, the control unit operates as described below:
The control unit executes the jog feed or incremental feed at a rapid traverse rate. The rapid traverse override is validated.
When the signal is switched from “1” to “0” or vice versa during jog feed or incremental feed, the feedrate is decelerated until it reaches zero, then increased to the specified value. During acceleration and deceleration, the feed axis and direction selection signal can be kept “1”.



⚠ WARNING
After the power is turned on, stored stroke check function does not work until the reference position return is completed. During this period, the control unit ignores manual rapid traverse selection signal RT, if it is set to “1”, and keeps moving the tool at a feedrate selected by the manual feedrate override signal. The bit 0 (RPD) of parameter No. 1401 can be set so the rapid traverse is validated before the reference position return is completed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn010	*JV7	*JV6	*JV5	*JV4	*JV3	*JV2	*JV1	*JV0
Gn011	*JV15	*JV14	*JV13	*JV12	*JV11	*JV10	*JV9	*JV8
Gn019	RT							
Gn100	+J8	+J7	+J6	+J5	+J4	+J3	+J2	+J1
Gn102	-J8	-J7	-J6	-J5	-J4	-J3	-J2	-J1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002								JAX

[Input type] Parameter input
[Data type] Bit path

#0 JAX Number of axes controlled simultaneously in jog feed, manual rapid traverse and manual reference position return
0: 1 axis
1: 3 axes

	#7	#6	#5	#4	#3	#2	#1	#0
1401								RPD

[Input type] Parameter input
[Data type] Bit path

#0 RPD Manual rapid traverse during the period from power-on time to the completion of the reference position return.
0: Disabled (Jog feed is performed.)
1: Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
1402				JRV			JOV	NPC

[Input type] Parameter input
[Data type] Bit path

#0 NPC Feed per revolution without the position coder (function for converting feed per revolution F to feed per minute F in the feed per revolution mode (G95)) is:
0: Not used
1: Used

#1 JOV Jog override is:
0: Enabled
1: Disabled (tied to 100%)

#4 JRV Jog feed or incremental feed is
0: Performed at feed per minute.
1: Performed at feed per rotation.

NOTE
1 Specify a feedrate in parameter No. 1423.
2 For the machining center system, the threading/synchronous feed is required.

1423	Jog feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
(1) When bit 4 (JRV) of parameter No. 1402 is set to 0 set jog feedrate (feed per minute) for each axis when manual feedrate override is 100%.
(2) When bit 4 (JRV) of parameter No. 1402 is set to 1set jog feedrate (feed per revolution) for each axis when manual feedrate override is 100%.

NOTE
This parameter is clamped to the axis-by-axis manual rapid traverse rate (parameter No. 1424).

1424	Manual rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rate of manual rapid traverse when the rapid traverse override is 100% for each axis.

NOTE
1 If 0 is set, the rate set in parameter No. 1420 (rapid traverse rate for each axis) is assumed.
2 When manual rapid traverse is selected (bit 0 (RPD) of parameter No. 1401 is set to 1), manual feed is performed at the feedrate set in this parameter, regardless of the setting of bit 4 (JRV) of parameter No. 1402.

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx				

[Input type] Parameter input
[Data type] Bit axis

#4 JGLx Acceleration/deceleration in jog feed
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied.
(Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)

1624	Time constant of acceleration/deceleration in jog feed for each axis.

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set the time constant used for acceleration/deceleration in jog feed for each axis.

1625	FL rate of acceleration/deceleration in jog feed for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set the FL rate of acceleration/deceleration in jog feed for each axis.
This parameter allows only the exponential type.

	#7	#6	#5	#4	#3	#2	#1	#0
3729							FPRs	

[Input type] Parameter input
[Data type] Bit spindle

#1 FPRs Feed per revolution (without a position coder) is:
0: Not used for a spindle.
1: Used for a spindle.

In a machine that does not use a position coder, by setting bit 1 (FPRs) of parameter No.3729 to 1for each axis, feed per revolution can be performed with a spindle command.
A feed per revolution is
specified with G95 (G99 for lathe systems) in the same way as for normal operation.
When multispindle control is performed, the target spindle for feed per revolution is selected with a position coder select signal (PC2SLC<Gn028.7>, PC3SLC<Gn026.0>, PC4SLC <Gn026.1>).

NOTE
Enable the constant surface speed control (bit 0 (SSC) of parameter No.8133 is 1).

	#7	#6	#5	#4	#3	#2	#1	#0
7001		JEX						

[Input type] Parameter input
[Data type] Bit path

#6 JEX The number of axes controlled simultaneously in jog feed, manual rapid traverse, and manual reference position return is:
0: Set by bit 0 (JAX) of parameter No. 1002.
1: The maximum number of simultaneously controlled axes.

	#7	#6	#5	#4	#3	#2	#1	#0
7103						HNT		

[Input type] Parameter input
[Data type] Bit path

#2 HNT When compared with the travel distance magnification selected by the manual handle feed travel distance selection signals (incremental feed signals) (MP1, MP2), the travel distance magnification for incremental feed/manual handle feed is:
0: Same.
1: 10 times greater.

Warning

⚠ **WARNING**
In incremental feed, when axis is diameter specification, travel distance is diameter value.

Note

NOTE
1 Time constant and method of automatic acceleration/deceleration for manual rapid traverse are the same as rapid traverse in programmed command (positioning by G00).
2 If a manual pulse generator is provided, the manual handle feed mode is enabled instead of incremental feed mode. However, using bit 0 (JHD) of parameter No. 7100 enables both manual handle and incremental feed in the manual handle feed mode.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Jog feed
	Incremental feed

3.2 MANUAL HANDLE FEED

Overview

In the handle mode, when the manual pulse generator on the machine operator's panel is rotated, the axis moves accordingly. Select the axis along which the tool is to be moved with the handle feed axis selection switches.
The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment.
The distance the tool is to be move can be multiplied by one of 4 magnifications selected by manual handle feed movement selection signals MP1, MP2<Gn019.4,5>. The distance can be further multiplied by 10 using bit 2 (HNT) of parameter No. 7103.
The number of manual pulse generators that can be attached is maximum 5. When using the fourth or fifth manual pulse generator, the function "Manual handle feed 4/5-unit" is needed.

NOTE
To use manual handle feed, set 1 in bit 0 (HPG) of parameter No.8131.

Explanation

- Manual handle feed in the jog feed mode

- Incremental feed in the manual handle feed mode

The following states can be selected using bit 0 (JHD) of parameter No. 7100.

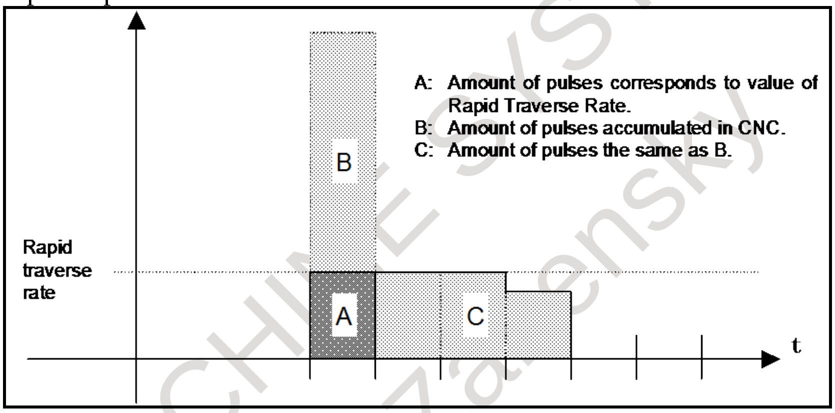
	JHD=0		JHD=1	
	Jog feed mode	**Manual handle feed mode**	**Jog feed mode**	**Manual handle feed mode**
Jog feed	Enabled	Disabled	Enabled	Disabled
Manual handle feed	Disabled	Enabled	Enabled	Enabled
Incremental feed	Disabled	Disabled	Disabled	Enabled

- Manual handle feed in TEACH IN JOG mode

By setting bit 1 (THD) of parameter No. 7100, manual handle feed in TEACH IN JOG mode can be enabled or disabled.

- When manual handle feed exceeding the rapid traverse rate is specified

The amount of pulses exceeding the rapid traverse rate can be saved by CNC as B. And amount of pulses B will be output as pulses C.



Amount of pulses output by CNC in Manual Handle Feed

For pulses exceeding rapid traverse rate, amount of pulses B is calculated in 2 cases as following:

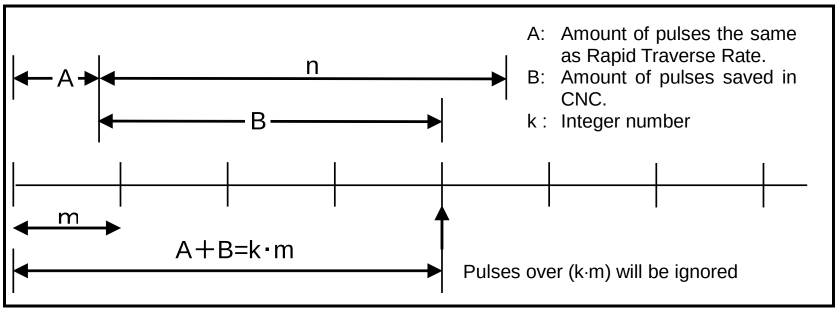
1) In case of parameter No.7117 = 0
The feedrate is clamped at the Rapid Traverse Rate and generated pulses exceeding the Rapid Traverse Rate are ignored (B=0).

2) In case of parameter No.7117 > 0
The feedrate is clamped as the Rapid Traverse Rate, but the pulses exceeding the Rapid Traverse Rate is not ignored. Amount of pulses accumulated in CNC is calculated as following. (Although the rotation of manual pulse generator is stopped, if there is pulses accumulated in CNC, it will be output and the tool will move as long as amount of it. Then the tool stops.)

Magnification set by manual handle feed movement selection signals MP1, MP2<Gn019.4,5> is m, value of parameter No.7117 is n.

n < m: Clamping is set performed at value of parameter No.7117.

n ≥ m: It is clamped in a range not exceeding n so that A+B shown in the following figure is an integral multiple of m. As a result, movement of the axis can be stopped at an integral multiple of the handle feed moving magnification.



Amount of pulses exceeding the Rapid Traverse Rate (n ≥ m)

> **NOTE**
> Due to change of mode, clamping can be performed not as an integral multiple of the selected magnification.
> The distance the tool moves may not match the graduations on the manual pulse generator.

- Upper limit of the feedrate in manual handle feed

The upper limit of the feedrate is determined as follows depending on the input signal (maximum manual handle feedrate switch signal HNDLF) from PMC.

- If HNDLF is “0”, clamping is performed at the manual rapid traverse rate (parameter No. 1424).
- If HNDLF is “1”, clamping is performed at the rate set in parameter No. 1434.

- Movement direction of an axis to the rotation of MPG

Bit 0 (HNGx) of parameter No. 7102 switches the direction of MPG in which the tool moves along an axis, corresponding to the direction in which the handle of the manual pulse generator is rotated.

Manual handle feed direction inversion signal HDN<Gn347.1> inverts the axis movement direction in relation to the rotation direction of the manual handle. The axis whose movement direction is to be inverted can be selected using bit 1 (HNAx) of parameter No. 7102.

The parameter and signal take effect only on the following functions:

- Manual handle feed
- Manual handle interruption

Warning

> ⚠ **WARNING**
> Rotating the handle quickly with a large magnification such as × 100 moves the tool too fast. The feedrate is clamped at the rapid traverse feedrate.

Note

> **NOTE**
> Rotate the manual pulse generator at a rate of five rotations per second or lower. If the manual pulse generator is rotated at a rate higher than five rotations per second, the tool may not stop immediately after the handle is no longer rotated or the distance the tool moves may not match the graduations on the manual pulse generator.

Signal

Manual Handle Feed Axis Selection Signals

HS1A to HS1D<Gn018.0 - 3>, HS1E<Gn411.0>,
HS2A to HS2D<Gn018.4 - 7>, HS2E<Gn411.1>,
HS3A to HS3D<Gn019.0 - 3>, HS3E<Gn411.2>,
HS4A to HS4D<Gn020.0 - 3>, HS4E<Gn411.3>,
HS5A to HS5D<Gn379.0 - 3>, HS5E<Gn412.0>

[Classification] Input signal

[Function] Selects the axis of manual handle feed. A set of five code signals, A, B, C, D and E is provided for each manual pulse generator. (Up to 5 generators can be used.) (For multi-path control, there is a set of code signals for each path for each manual pulse generator.) The number in the signal name indicates the number of the manual pulse generator to be used.

HS1A

1 : Selection of axis to be moved by 1st. manual pulse generator
2 : Selection of axis to be moved by 2nd. manual pulse generator
3 : Selection of axis to be moved by 3rd. manual pulse generator
4 : Selection of axis to be moved by 4th. manual pulse generator
5 : Selection of axis to be moved by 5th. manual pulse generator

Code signals A, B, C, D, and E correspond to the feed axes listed in following table:

Manual handle feed axis selection signals					Feed axis
HSnE	HSnD	HSnC	HSnB	HSnA	
0	0	0	0	0	No selection (None of axis is fed)
0	0	0	0	1	1st axis
0	0	0	1	0	2nd axis
0	0	0	1	1	3rd axis
		to			to
1	0	1	1	0	22nd axis
1	0	1	1	1	23rd axis
1	1	0	0	0	24th axis

Manual handle feed amount selection signals (Incremental feed signals)

MP1, MP2<Gn019.4,5>, MP21, MP22 <Gn087.0,1>, MP31, MP32 <Gn087.3,4>,
MP41, MP42 <Gn087.6,7>, MP51, MP52 <Gn380.0,1>

[Classification] Input signal

[Function] This signal selects the distance traveled per pulse from the manual pulse generator during the manual handle feed or manual handle interrupt. It also selects the distance traveled per incremental feed step.

The table below lists the signal-to-distance correspondence.

Manual Handle Feed Amount Selection signals		Distance Traveled		
MP2	MP1	Manual Handle Feed	Manual Handle Interruption	Incremental Feed
0	0	Minimum unit × 1	Minimum unit × 1	Minimum unit × 1
0	1	Minimum unit × 10	Minimum unit × 10	Minimum unit × 10
1	0	Minimum unit × m[*1]	Minimum unit × m[*1]	Minimum unit × 100
1	1	Minimum unit × n[*1]	Minimum unit × n[*1]	Minimum unit × 1000

*1 Scale factors m and n are specified using parameter No.7113 and 7114.

Scale factors m and n of each axis are specified using parameter No.12350 and 12351.

> ⚠ **CAUTION**
> 1 Because the least input increment is used as the units for manual handle and incremental feed, the same value represents a different distance depending on whether the metric or inch input system is used.
> 2 For an axis under diameter programming, the tool moves by the diameter value.

Furthermore, Manual Handle Feed Amount Selection Signals of each Manual Pulse Generator can be used by setting bit 5 (MPX) of parameter No.7100. The relation between Manual Handle Feed Amount Selection signals of each Manual Pulse Generator and parameter to set magnification is showed in the next table.

Value of bit 5 (MPX) of parameter No. 7100	Manual Pulse Generator	Manual Handle Feed Amount Selection signals	Parameter of Magnification	
			m_x	n_x
MPX=0	1st.-5th. Manual Pulse Generator	MP1,MP2	No.7113	No.7114
MPX=1	1st. Manual Pulse Generator	MP1,MP2	No.7113	No.7114
	2nd. Manual Pulse Generator	MP21,MP22	No.7131	No.7132
	3rd. Manual Pulse Generator	MP31,MP32	No.7133	No.7134
	4th. Manual Pulse Generator	MP41,MP42	No.7135	No.7136
	5th. Manual Pulse Generator	MP51,MP52	No.7137	No.7138

Maximum manual handle feedrate switch signal HNDLF<Gn023.3>

[Classification] Input signal

[Function] Selects the maximum feedrate in manual handle feed. The maximum feedrate in manual handle feed is determined according to the state of this signal as follows.

Maximum manual handle feedrate switch signal	Maximum feedrate
0	Manual rapid traverse rate (parameter No. 1424)
1	Maximum manual handle feedrate (parameter No. 1434)

Manual handle feed direction inversion signal HDN<Gn347.1>

[Classification] Input signal

[Function] Inverts the rotation direction of the manual pulse generator and the axis movement direction in manual handle feed.

0: Does not invert the axis movement direction in relation to the rotation direction of the manual pulse generator.

1: Inverts the axis movement direction in relation to the rotation direction of the manual pulse generator.

The axis whose movement direction is to be inverted by this signal can be selected using bit 1 (HNAx) of parameter No. 7102. This signal does not work on the rotation direction of manual linear/circular interpolation.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn018	HS2D	HS2C	HS2B	HS2A	HS1D	HS1C	HS1B	HS1A
Gn019			MP2	MP1	HS3D	HS3C	HS3B	HS3A
Gn020					HS4D	HS4C	HS4B	HS4A
Gn023					HNDLF			
Gn087	MP42	MP41		MP32	MP31		MP22	MP21
Gn347							HDN	
Gn379					HS5D	HS5C	HS5B	HS5A
Gn380							MP52	MP51
Gn411					HS4E	HS3E	HS2E	HS1E
Gn412								HS5E

Parameter

1424	Manual rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set the rate of manual rapid traverse when the rapid traverse override is 100% for each axis.

NOTE
1. If 0 is set, the rate set in parameter No. 1420 (rapid traverse rate for each axis) is assumed.
2. When manual rapid traverse is selected (bit 0 (RPD) of parameter No. 1401 is set to 1), manual feed is performed at the feedrate set in this parameter, regardless of the setting of bit 4 (JRV) of parameter No. 1402.

1434	Maximum manual handle feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set a maximum manual handle feedrate for each axis in case of maximum manual handle feedrate switch signal HNDLF<Gn023.3>=1.

	#7	#6	#5	#4	#3	#2	#1	#0
7100			MPX				THD	JHD

[Input type] Parameter input
[Data type] Bit path

#0 JHD Manual handle feed in JOG feed mode or incremental feed in the manual handle feed is:
0: Invalid.
1: Valid.

#1 THD In the TEACH IN JOG mode, the manual pulse generator is:
0: Disabled.
1: Enabled.

#5 MPX In Manual handle feed mode:
0 : Manual handle feed amount selection signals MP1 and MP2 <Gn019.4 and Gn019.5> for the 1st manual pulse generator are used as signals common to all manual pulse generators.
1 : Manual handle feed amount selection signals differ depending on the manual pulse generator as follow:
1st. Manual Pulse Generator : MP1, MP2 <Gn019.4, Gn019.5>
2nd. Manual Pulse Generator : MP21, MP22 <Gn087.0, Gn087.1>
3rd. Manual Pulse Generator : MP31, MP32 <Gn087.3, Gn087.4>
4th. Manual Pulse Generator : MP41, MP42 <Gn087.6, Gn087.7>
5th. Manual Pulse Generator : MP51, MP52 <Gn380.0, Gn380.1>

	#7	#6	#5	#4	#3	#2	#1	#0
7102							HNAx	HNGx

[Input type] Parameter input
[Data type] Bit axis

#0 HNGx Axis movement direction for rotation direction of manual pulse generator
0: Same in direction
1: Reverse in direction

#1 HNAx When manual handle feed direction inversion signal HDN <Gn0347.1> is set to "1", the direction of movement is set for each axis with respect to the rotation direction of the manual pulse generator.
0: The axis movement direction is the same as the direction in which the manual pulse generator rotates.
1: The axis movement direction is opposite to the direction in which the manual pulse generator rotates.
When the rotation direction is reversed by manual handle feed direction inversion signal HDN <Gn0347.1>, the rotation axis direction obtained by the setting of bit 0 (HNGx) of parameter No. 7102 is reversed.

	#7	#6	#5	#4	#3	#2	#1	#0
7103						HNT		

[Input type] Parameter input
[Data type] Bit path

#2 HNT When compared with the travel distance magnification selected by the manual handle feed travel distance selection signals (incremental feed signals MP1, MP2), the travel distance magnification for incremental feed/manual handle feed is:

0: Same.
1: 10 times greater.

	#7	#6	#5	#4	#3	#2	#1	#0
7105							HDX	

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 HDX Manual handle for I/O Link connection is:

0: Automatically set.
1: Manually set.

> **NOTE**
> In manual setting, parameters Nos. 12300 to 12304 and 12340 to 12344 must be set by manual to connect Manual Pulse Generator with I/O Link.

7113	Manual handle feed magnification m

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 2000

This parameter sets the magnification m when manual handle feed movement selection signals MP1 and MP2 are set to "0" and "1".

7114	Manual handle feed magnification n

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 2000

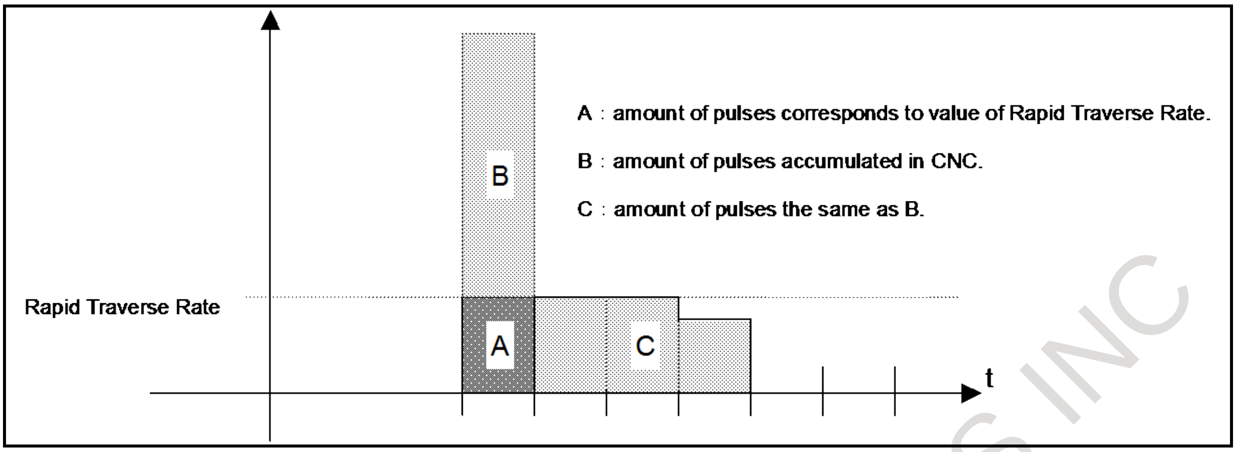
This parameter sets the magnification when manual handle feed movement selection signals MP1 and MP2 are set to "1".

7117	Allowable number of pulses that can be accumulated during manual handle feed

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Pulse
[Valid data range] 0 to 999999999

This parameter sets the number of pulses from the manual pulse generator that exceed the rapid traverse rate and can be accumulated without being discarded if manual handle feed faster than the rapid traverse rate is specified.

The amount of pulses exceeding the rapid traverse rate can be saved by CNC as B. And amount of pulses B will be exported as pulses C.



Amount of pulses exported by CNC in Manual Handle Feed

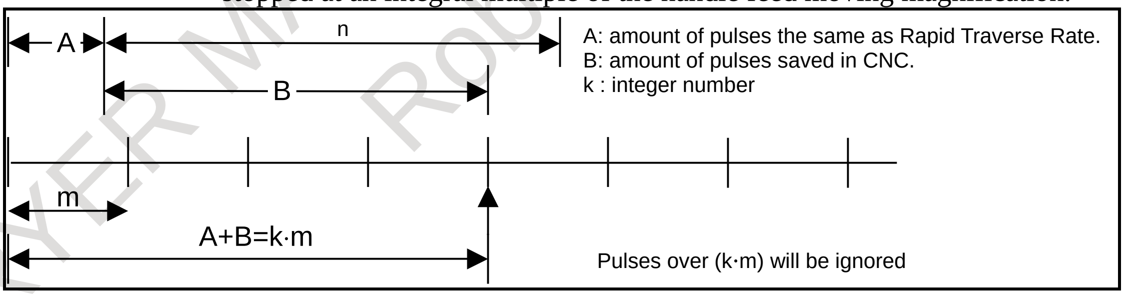
Amount of pulses B is calculated in 2 cases as following:

1) In case of parameter No.7117 = 0
   The feedrate is clamped at the Rapid Traverse Rate and generated pulses exceeding the Rapid Traverse Rate are ignored (B=0)
2) In case of parameter No.7117 > 0
   The feedrate is clamped as the Rapid Traverse Rate, but the handle pulses exceeding the Rapid Traverse Rate is not ignored. Amount of pulses accumulated in CNC is calculated as following. (Although the rotation of manual pulse generator is stopped, if there is pulses accumulated in CNC, it will be output and the tool will move as long as amount of it. Then the tool stops.)

Magnification set by manual handle feed amount selection signals MP1, MP2 <Gn019.4, Gn019.5> is m, value of parameter No.7117 is n.

n < m: Clamping is set performed at value of parameter No.7117.

n ≥ m: It is clamped in a range not exceeding n so that A+B shown in the following figure is an integral multiple of m. As a result, movement of the axis can be stopped at an integral multiple of the handle feed moving magnification.



Amount of pulses exceeding the Rapid Traverse Rate (n ≥ m)

> **NOTE**
> Due to change of mode, clamping can be performed not as an integral multiple of the selected magnification.
> The distance the tool moves may not match the graduations on the manual pulse generator.

7131	**Manual handle feed magnification m2 / 2nd. manual pulse generator**
7132	**Manual handle feed magnification n2 / 2nd. manual pulse generator**
7133	**Manual handle feed magnification m3 / 3rd. manual pulse generator**

7134	**Manual handle feed magnification n3 / 3rd. manual pulse generator**
7135	**Manual handle feed magnification m4 / 4th. manual pulse generator**
7136	**Manual handle feed magnification n4 / 4th. manual pulse generator**
7137	**Manual handle feed magnification m5 / 5th. manual pulse generator**
7138	**Manual handle feed magnification n5 / 5th. manual pulse generator**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 2000

The 'mx' sets magnification when manual handle feed amount selection signals MPx1 = 0, MPx2 = 1. The 'nx' sets magnification when manual handle feed amount selection signals MPx1 = 1, MPx2 = 1.

12300	**X address of the 1st. manual pulse generator**
12301	**X address of the 2nd. manual pulse generator**
12302	**X address of the 3rd. manual pulse generator**
12303	**X address of the 4th. manual pulse generator**
12304	**X address of the 5th. manual pulse generator**

> **NOTE**
> When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] -1, 0 to 127, 200 to 327, 400 to 527, 600 to 727

To set X address of manual pulse generator connected with I/O Link in PMC.
When the manual pulse generator is not connected, set -1 to this parameter.

	PMC Path	X Address
1st. manual pulse generator	No. 12340	No. 12300
2nd. manual pulse generator	No. 12341	No. 12301
3rd. manual pulse generator	No. 12342	No. 12302
4th. manual pulse generator	No. 12343	No. 12303
5th. manual pulse generator	No. 12344	No. 12304

Setting of PMC path set with parameters Nos.12340 to 12344 is as follows.

Value	PMC Path
0	1st. PMC
1	
2	2nd. PMC
3	3rd. PMC

> **NOTE**
> Set these parameters when bit 1 (HDX) of parameter No. 7105 is set to 1. When HDX = 0, these parameters are automatically set. If a manual handle is not connected when HDX = 0, -1 is set automatically.

12340	PMC path of the 1st. manual pulse generator connected with I/O Link
12341	PMC path of the 2nd. manual pulse generator connected with I/O Link
12342	PMC path of the 3rd. manual pulse generator connected with I/O Link
12343	PMC path of the 4th. manual pulse generator connected with I/O Link
12344	PMC path of the 5th. manual pulse generator connected with I/O Link

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 3
Referring to parameters Nos. 12300 to 12304.

12350	Manual handle feed magnification m in each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 2000
For each axis, this parameter sets the magnification m when manual handle feed movement selection signals MP1 = 0, MP2 = 1.

NOTE
When value is set to 0 for this parameter, the parameter No. 7113 is valid.

12351	Manual handle feed magnification n in each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 2000
For each axis, this parameter sets the magnification when manual handle feed movement selection signals MP1 = “1”, MP2 = “1”.

NOTE
When value is set to 0 for this parameter, the parameter No. 7114 is valid.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Manual handle feed

3.2.1 Jog and Handle Simultaneous Mode

Overview

Usually, manual handle feed is enabled only when the operation mode is set to the manual handle feed mode. By setting bit 0 (JHD) of parameter No. 7100, however, manual handle feed can be performed even in Jog feed mode. At this setting, moreover, incremental feed can be performed even in the manual handle feed mode.

Explanation

When bit 0 (JHD) of parameter No. 7100 is set to 1 (jog and handle simultaneous mode), the enabled feed in each mode is as follows.

	Jog feed	Manual handle feed	Incremental feed
Jog feed mode (JOG mode)	Enabled	Enabled	Disabled
Manual handle feed mode (HND mode)	Disabled	Enabled	Enabled

- Superimposed feed of the jog feed and the manual handle feed

When bit 0 (JHD) of parameter No. 7100 is set to 1 (jog and handle simultaneous mode), the jog feed and the manual handle feed can be superimposed in jog feed mode. In order to perform superimposed operation, type of acceleration/deceleration and the time constant are the same in the jog feed and the manual handle feed. The type of acceleration/deceleration and the time constant in manual handle feed follow the type of acceleration/deceleration and the time constant in previous commanded feed (either jog feed or manual rapid traverse).

Fig. 3.2.1 (a) and (b) show the following operation in JOG mode.
(1) Jog feed
(2) Jog feed and manual handle feed
When the manual rapid traverse selection signal RT<Gn019.7> is set to "0", the type of acceleration/deceleration and the time constant in jog feed are the same as those in jog feed. The type of acceleration/deceleration and the time constant in manual handle feed is the same as those in previous commanded jog feed.

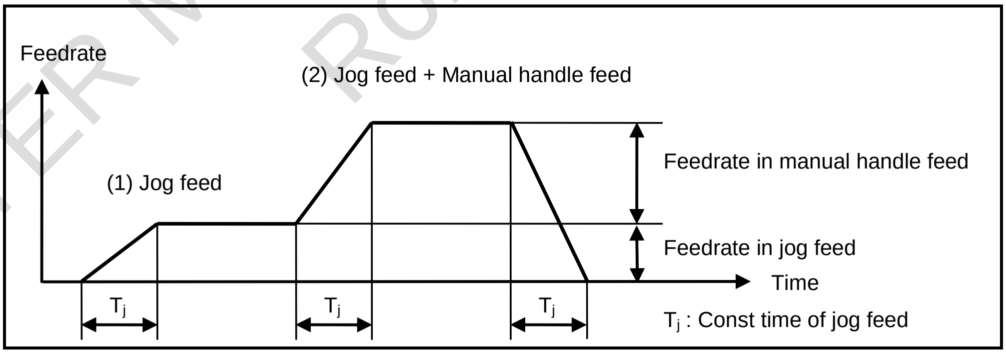


Fig. 3.2.1 (a) Superimposed operation (when RT<Gn019.7> is "0")

When the manual rapid traverse selection signal RT<Gn019.7> is set to "1", the type of acceleration/deceleration and the time constant in jog feed are the same as those in rapid traverse. The type of acceleration/deceleration and the time constant in manual handle feed is the same as those in previous commanded manual rapid traverse.

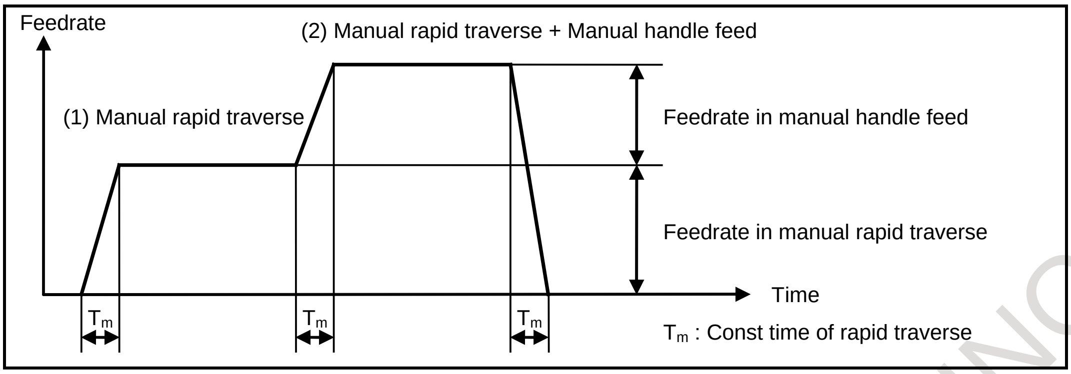


Fig. 3.2.1 (b) Superimposed operation (when RT<Gn019.7> is "1")
(Acceleration/deceleration of time fixed type is used for positioning of linear interpolation type.)

Fig. 3.2.1 (c) and (d) show the operation which commanded in the following order in JOG mode.
(1) Manual handle feed
(2) Jog feed
(3) Manual handle feed
The type of acceleration/deceleration and the time constant in the manual handle feed are switched after the jog feed.

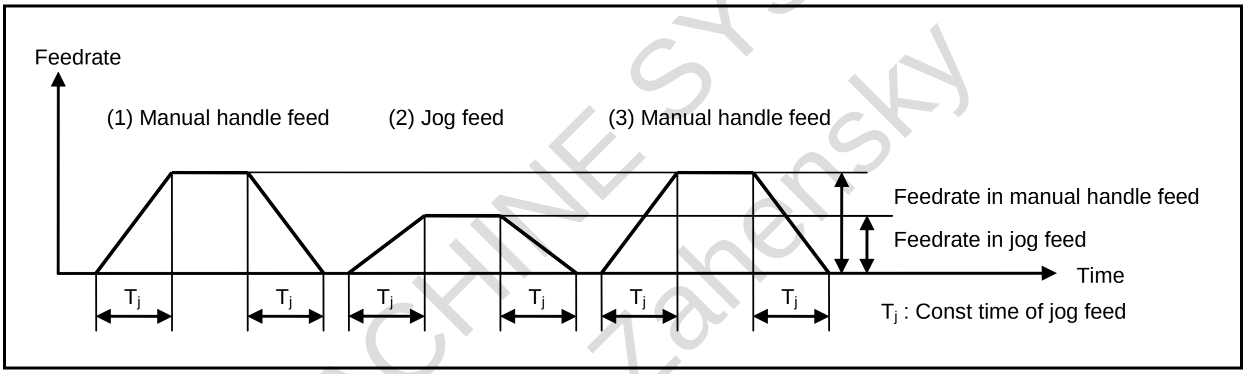


Fig. 3.2.1 (c) Switch of the jog feed and the manual handle feed (when RT<Gn019.7> is "0")

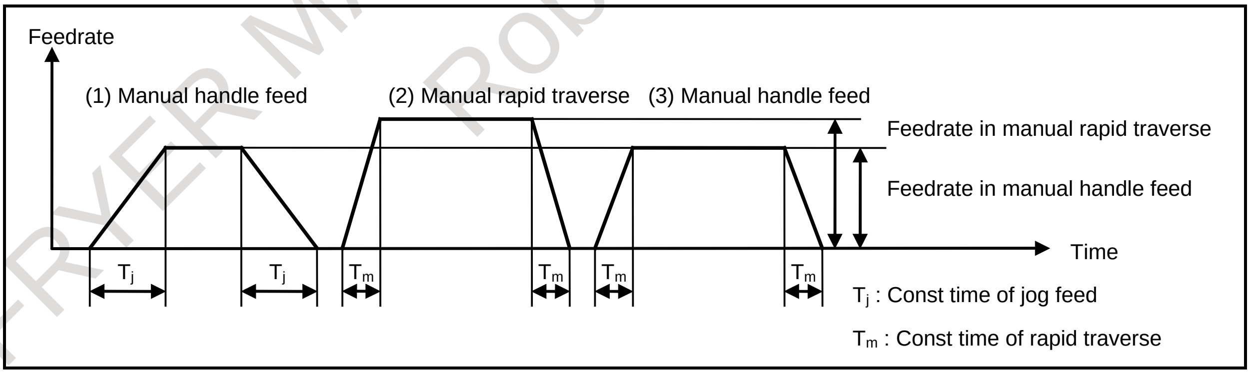


Fig. 3.2.1 (d) Switch of the jog feed and the manual handle feed (when RT<Gn019.7> is "1")

Fig.3.2.1 (e)and (f) show the operation which started (2) manual handle feed immediately after (1) jog feed. Feed can be switched without deceleration stop because the same type of acceleration/deceleration and time constant are used.

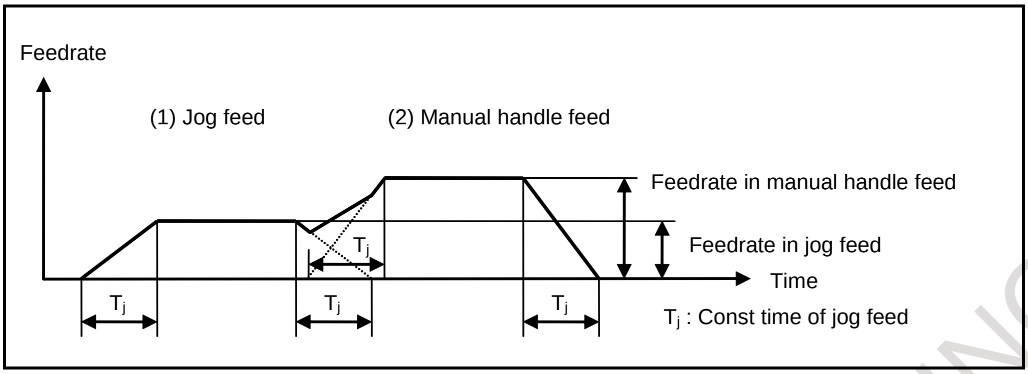


Fig.3.2.1 (e) Switch from the jog feed to the manual handle feed (when RT<Gn019.7> is "0")

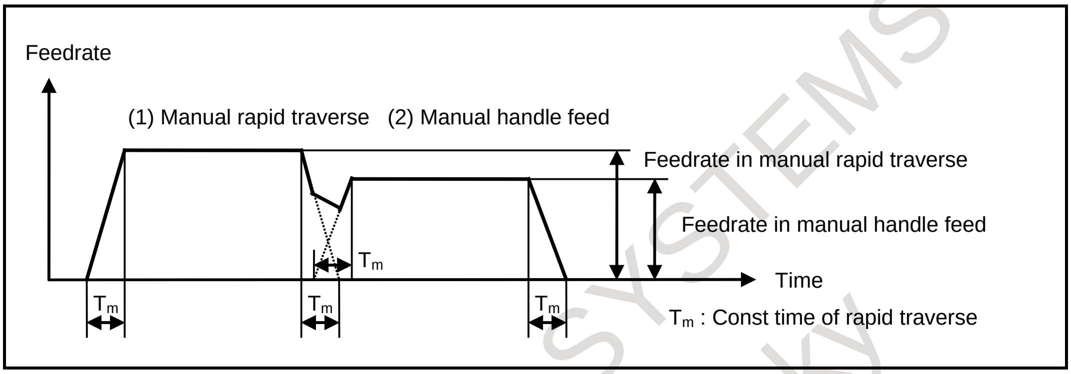


Fig.3.2.1 (f) Switch from the jog feed to the manual handle feed (when RT<Gn019.7> is "1")

- Prohibition of superimposed feed of the jog feed and the manual handle feed (Bit 1 (FJH) of parameter No. 7107)

When bit 1 (FJH) of parameter No. 7107 is set to 1, the jog feed and the manual handle feed are not superimposed. Priority can be placed on the jog feed if both are executed at the same time.

Fig. 3.2.1 (g) and (h) show the operation which commanded in the following order in JOG mode.
(1) Manual handle feed
(2) Jog feed
(3) Manual handle feed
When manual rapid traverse selection signal RT <Gn019.7> is set to "0", the type of acceleration/deceleration and the time constant in jog feed and manual handle feed are those of jog feed.

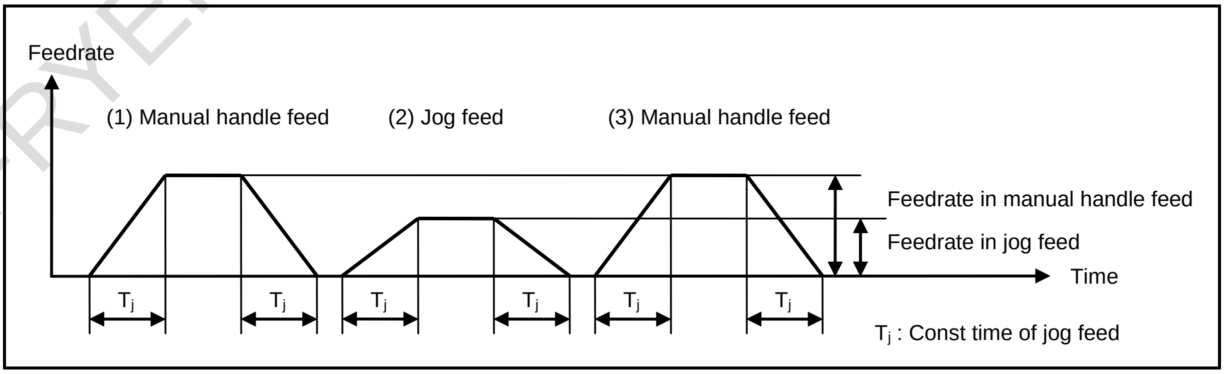


Fig. 3.2.1 (g) Switch of the jog feed and the manual handle feed (when RT<Gn019.7> is "0")

When manual rapid traverse selection signal RT <Gn019.7> is set to "1", the type of acceleration/deceleration and the time constant in jog feed are those of rapid traverse, the type of acceleration/deceleration and the time constant in manual handle feed are those of jog feed.

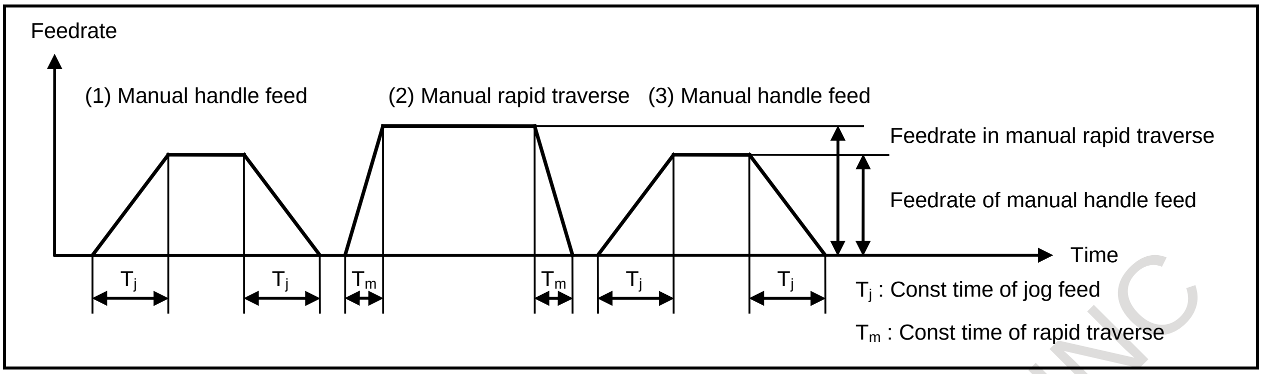


Fig. 3.2.1 (h) Switch of the jog feed and the manual handle feed (when RT<Gn019.7> is "1")

Fig.3.2.1 (i) and (j) show the operation which started (2) manual handle feed immediately after (1) jog feed.
The type of acceleration/deceleration and the time constant are switched after checking that delay in acceleration/deceleration is 0.

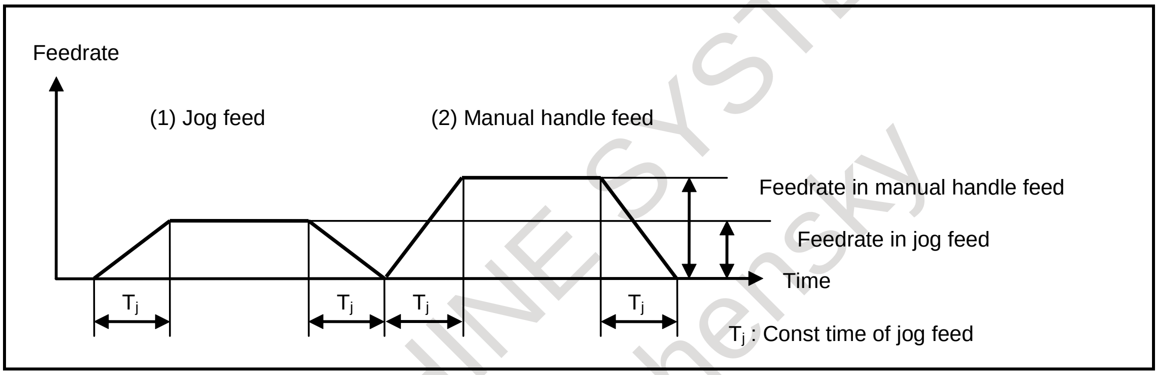


Fig.3.2.1 (i) Switch from the jog feed to the manual handle feed (when RT<Gn019.7> is "0")

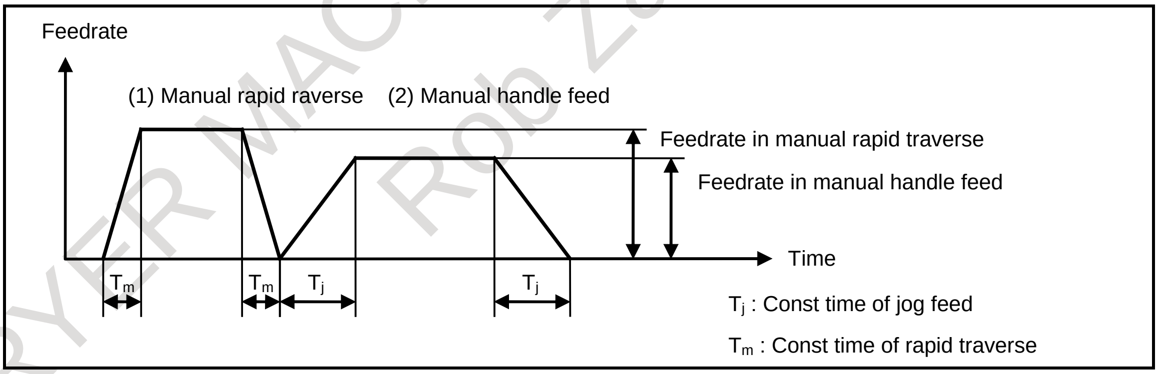


Fig.3.2.1 (j) Switch from the jog feed to the manual handle feed (when RT<Gn019.7> is "1")

- TEACH IN JOG mode

When bit 1 (THD) of parameter No.7100 is set to 1, manual handle can be effective in TEACH IN JOG mode. Bit 1 (FJH) of parameter No.7107 is effective in this case.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7100							THD	JHD

[Input type] Parameter input
[Data type] Bit path

#0 JHD Manual handle feed in JOG feed mode or incremental feed in the manual handle feed is:
0: Invalid.
1: Valid.

#1 THD In the TEACH IN JOG mode, the manual pulse generator is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
7107							FJH	

[Input type] Parameter input
[Data type] Bit

#1 FJH When bit 0 (JHD) of parameter No. 7100 is 1, jog feed and manual handle feed in jog feed mode, or incremental feed and manual handle feed in manual handle feed are:
0: superimposed. The type of acceleration/deceleration and the time constant in manual handle feed are the type of acceleration/deceleration and the time constant in previous commanded feed (either jog feed or manual rapid traverse).
1: not superimposed. Priority can be placed on jog feed or incremental feed if they and manual handle feed are executed at the same time. If there is an axis for jog feed or incremental feed even for one axis in the path, handle feed is invalid for all axes in the path. The type of acceleration/deceleration and the time constant in manual handle feed are always those of jog feed.

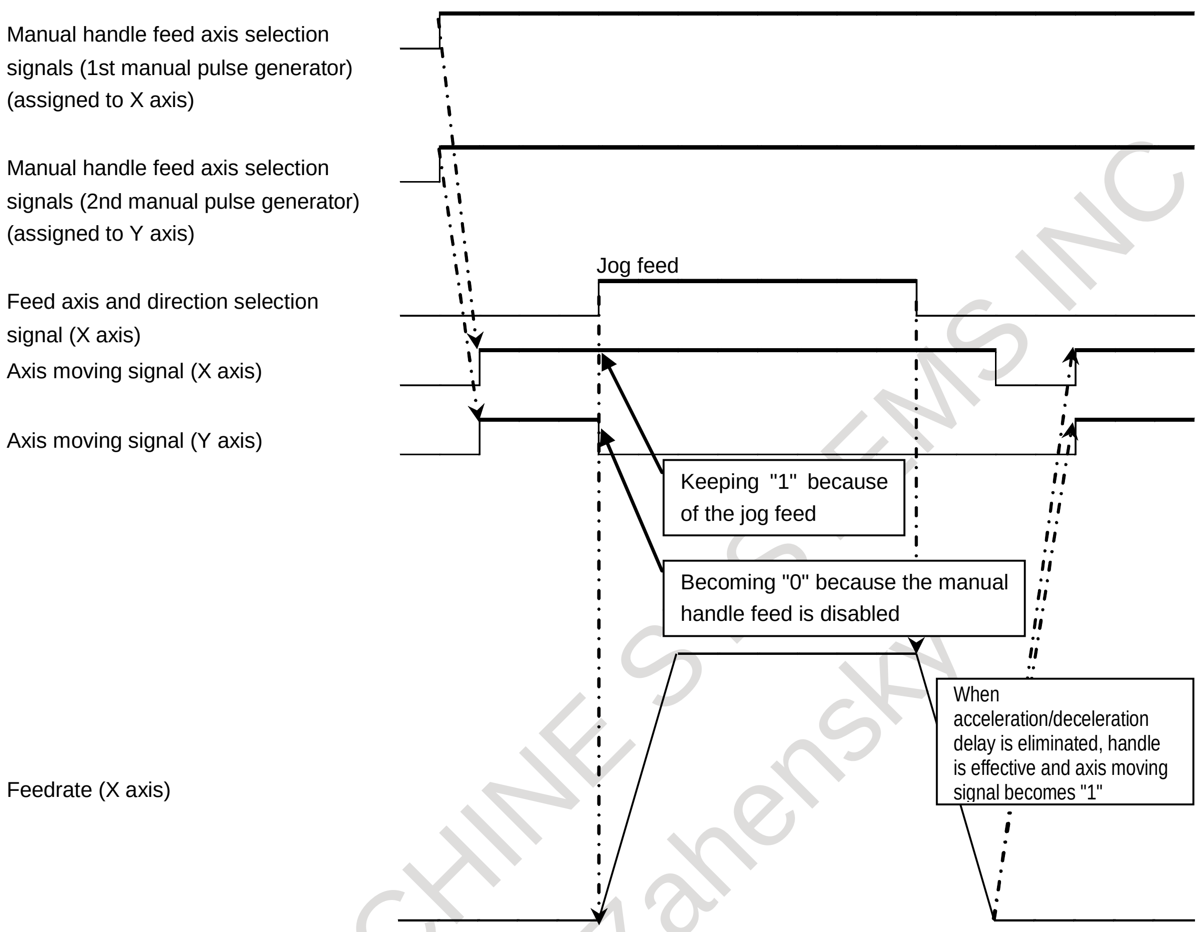
> **NOTE**
> When bit 1 (THD) of parameter No.7100 is set to 1, this parameter is effective to jog feed and manual handle feed in TEACH IN JOG mode.

Notes

1) Axis moving signals MVx<Fn102> of axes that manual handle feed is effective become "1". When bit 1 (FJH) of parameter No. 7107 is set to 1, even if the manual handle is allocated in axes other than the jog feed axes, the signals MVx<Fn102> of those axes does not become "1" because manual handle feed of those axes is not effective. When the manual handle feed become effective after checking that delay in acceleration/deceleration is 0, the signals MVx<Fn102> become "1".

Example of timing diagram of MVx<Fn102> when bit 1 (FJH) of parameter No. 7107 is 1

Jog feed with X axis is executed when the manual handle is allocated in X axis and Y axis.



*Above-mentioned timing that Axis moving signal becomes "1" after jog feed is one example.
That is different according to the specified speed and the in-position width.

3.3 MANUAL HANDLE INTERRUPT

Overview

In the automatic operation mode (manual data input, DNC operation, or memory operation) or memory edit mode, by rotating the manual pulse generator, handle feed can be performed by superimposing on the movement by automatic operation. The axis to which the handle interrupt is applied is selected using the manual handle interrupt axis selection signal. The minimum travel distance per graduation is the least command increment. The distance can be scaled by one of four types of magnification that is selected using manual handle feed movement selection signals MP1 and MP2 <Gn019 .4 and .5>.
The scaled distance can be multiplied by 10 using bit 3 (HIT) of parameter No. 7103. Each scale factor can be selected using the manual handle travel distance selection signals (refer to "Manual Handle Feed").

> ⚠ **WARNING**
> The amount of movement per graduation for manual handle interrupt is equal to the least input increment like that for manual handle feed. For example, for metric input on an inch machine, the tool moves by 0.01 inches for 254 graduations; for inch input on a millimeter machine, it moves by 0.254 mm for 100 graduations.

Signal

Manual Handle Interrupt Axis Selection Signals

HS1IA to HS1ID<Gn041.0 to 3>, HS1IE<Gn411.4>
HS2IA to HS2ID<Gn041.4 to 7>, HS2IE<Gn411.5>
HS3IA to HS3ID<Gn042.0 to 3>, HS3IE<Gn411.6>
HS4IA to HS4ID<Gn088.4 to 7>, HS4IE<Gn411.7>
HS5IA to HS5ID<Gn379.4 to 7>, HS5IE<Gn412.4>

[Classification] Input signal

[Function] These signals select an axis to which the manual handle interrupt is applied. There are five sets of signals, each corresponding to a manual pulse generator (up to five). Each set consists of five code signals A, B, C, D and E. The number in the signal name indicates the number of the manual pulse generator to be used.

HSxIA

x : 1 Selects the axis for which manual pulse generator No. 1 is used
2 Selects the axis for which manual pulse generator No. 2 is used
3 Selects the axis for which manual pulse generator No. 3 is used
4 Selects the axis for which manual pulse generator No. 4 is used
5 Selects the axis for which manual pulse generator No. 5 is used

The correspondence between code signals A, B, C, D, and E and selected feed axes is similar to the correspondence of the manual handle feed axis selection signals. See "Manual Handle Feed Axis Selection signals."

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn041	HS2ID	HS2IC	HS2IB	HS2IA	HS1ID	HS1IC	HS1IB	HS1IA
Gn042					HS3ID	HS3IC	HS3IB	HS3IA
Gn088	HS4ID	HS4IC	HS4IB	HS4IA				
Gn379	HS5ID	HS5IC	HS5IB	HS5IA				
Gn411	HS4IE	HS3IE	HS2IE	HS1IE				
Gn412				HS5IE				

Parameter

1466	Feedrate for retraction in threading cycle G92 or G76
	Feedrate for retraction in threading cycle G76.7

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
When threading cycle G92, G76 or G76.7 is specified, retraction is performed after threading. Set a feedrate for this retraction.

> ⚠ **WARNING**
> When the manual handle interruption is valid, set the same value as the parameter No.1430 to the parameter No.1466.

> **NOTE**
> When this parameter is set to 0 or bit 1 (CFR) of parameter No. 1611 is set to 1, the rapid traverse rate set in parameter No. 1420 is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1606								MNJx

[Input type] Parameter input
[Data type] Bit axis

#0 MNJx In manual handle interrupt or automatic manual simultaneous operation (interrupt type):
0: Only cutting feed acceleration/deceleration is enabled, and jog feed acceleration/deceleration is disabled.
1: Both cutting feed acceleration/deceleration and jog feed acceleration/deceleration are applied.

	#7	#6	#5	#4	#3	#2	#1	#0
6004						VHD		

[Input type] Parameter input
[Data type] Bit path

#2 VHD With system variables #5121 to #5140:
0: The tool offset value (geometry offset value) in the block currently being executed is read. (This parameter is valid only when tool geometry/tool wear compensation memories are available.)
1: An interrupt travel distance based on manual handle interrupt is read.

	#7	#6	#5	#4	#3	#2	#1	#0
7100					HCL			

[Input type] Parameter input
[Data type] Bit path

#3 HCL The clearing of handle interruption amount display by soft key [INTRPT CANCEL] operation is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
7102								HNGx

[Input type] Parameter input
[Data type] Bit axis

#0 HNGx Axis movement direction for rotation direction of manual pulse generator
0: Same in direction
1: Reverse in direction

	#7	#6	#5	#4	#3	#2	#1	#0
7103					HIT		RTH	

[Input type] Parameter input
[Data type] Bit path

#1 RTH By a reset or emergency stop, the amount of manual handle interruption is:
0: Not canceled.
1: Canceled.

#3 HIT When compared with the travel distance magnification selected by the manual handle feed travel distance selection signals (incremental feed signals MP1, MP2 <Gn019.4, Gn019.5>), the travel distance magnification for manual handle interrupt is:
0: Same.
1: 10 times greater.

7113	Manual handle feed magnification m

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 2000
This parameter sets the magnification m when manual handle feed movement selection signals MP1 and MP2 are set to “0” and “1”.

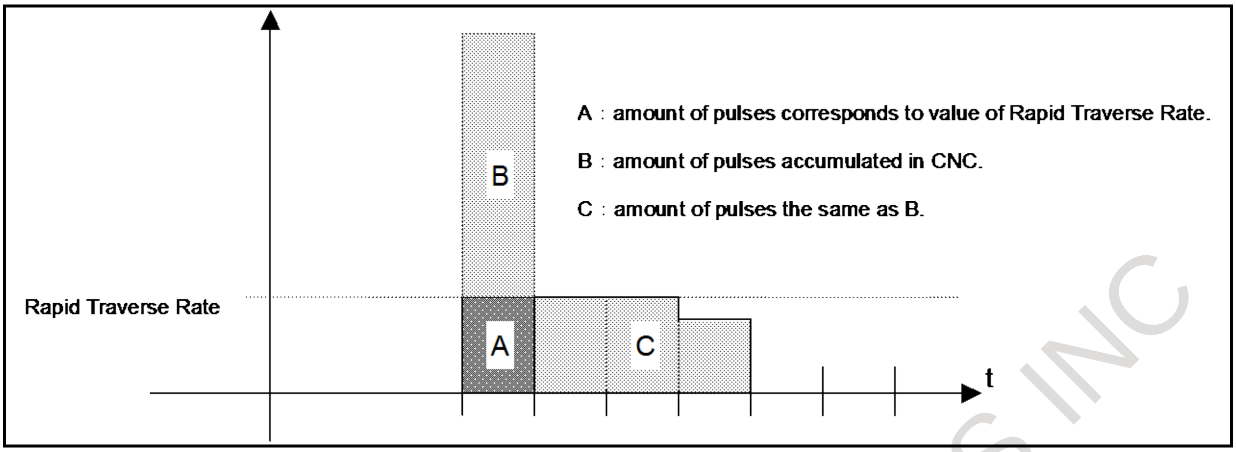
7114	Manual handle feed magnification n

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 2000
This parameter sets the magnification when manual handle feed movement selection signals MP1 and MP2 are set to “1”.

7117	Allowable number of pulses that can be accumulated during manual handle feed

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Pulse
[Valid data range] 0 to 999999999
This parameter sets the number of pulses from the manual pulse generator that exceed the rapid traverse rate and can be accumulated without being discarded if manual handle feed faster than the rapid traverse rate is specified.

The amount of pulses exceeding the rapid traverse rate can be saved by CNC as B. And amount of pulses B will be exported as pulses C.



Amount of pulses exported by CNC in Manual Handle Feed

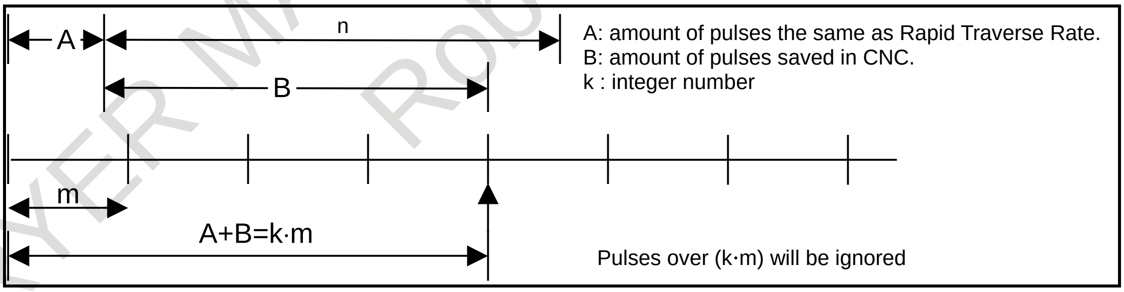
Amount of pulses B is calculated in 2 cases as following:

1) In case of parameter No.7117 = 0
   The feedrate is clamped at the Rapid Traverse Rate and generated pulses exceeding the Rapid Traverse Rate are ignored (B=0)
2) In case of parameter No.7117 > 0
   The feedrate is clamped as the Rapid Traverse Rate, but the pulses exceeding the Rapid Traverse Rate is not ignored. Amount of pulses accumulated in CNC is calculated as following. (Although the rotation of manual pulse generator is stopped, if there is pulses accumulated in CNC, it will be output and the tool will move as long as amount of it. Then the tool stops.)

Magnification set by manual handle feed amount selection signals MP1, MP2 <Gn019.4, Gn019.5> is m, value of parameter No.7117 is n.

n < m: Clamping is set performed at value of parameter No.7117.

n ≥ m: It is clamped in a range not exceeding n so that A+B shown in the following figure is an integral multiple of m. As a result, movement of the axis can be stopped at an integral multiple of the handle feed moving magnification.



Amount of pulses exceeding the Rapid Traverse Rate (n ≥ m)

NOTE
Due to change of mode, clamping can be performed not as an integral multiple of the selected magnification.
The distance the tool moves may not match the graduations on the manual pulse generator.

	#7	#6	#5	#4	#3	#2	#1	#0
10480							**RMI**	

[Input type] Parameter input
[Data type] Bit path

#1 RMI Manual handle interrupt for an axis specified in rapid traverse (G00) mode is:
0: Disabled.
1: Enabled.

> **NOTE**
> 1 Travel distance of manual handle interrupt depends on the setting of allowable number of pulses that can be accumulated during manual handle feed (parameter No.7117)
> 2 During axis movement at rapid traverse rate, axis movement by manual handle interrupt is output during deceleration so as not to exceed rapid traverse rate.

12350	Manual handle feed magnification m in each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 2000
For each axis, this parameter sets the magnification m when manual handle feed movement selection signals MP1 = “0”, MP2 = “1”.

> **NOTE**
> When value is set to 0 for this parameter, the parameter No. 7113 is valid.

12351	Manual handle feed magnification n in each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 2000
For each axis, this parameter sets the magnification when manual handle feed movement selection signals MP1 = “1”, MP2 = “1”.

> **NOTE**
> When value is set to 0 for this parameter, the parameter No. 7114 is valid.

Warning

> ⚠ **WARNING**
> When the manual handle interruption is valid, set the same value as the parameter No.1430 to the parameter No.1466.

Note

> **NOTE**
> 1 In the manual operation mode (such as the jog feed mode, manual handle feed mode, or TEACH IN HANDLE mode), handle interrupt is disabled.
> 2 When the machine lock or interlock is on, the tool is not moved by handle interrupt.
> 3 Manual handle interrupt is not effective for an axis in either of the following states:
> - Follow-up
> - PMC axis control

> **NOTE**
> 4 Manual handle interrupt is not allowed for an axis specified in G00 mode. But manual handle interrupt is enabled within the rapid traverse rate of the specified axis by setting bit 1 (RMI) of parameter No.10480.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Manual handle interrupt
CONNECTION MANUAL (FUNCTION) (B-64693EN-1) (This manual)	Manual handle feed

3.3.1 Manual Interruption of 3-dimensional Coordinate System Conversion

Overview

When the manual pulse generator is rotated in the 3-dimensional coordinate conversion mode, the travel distance specified by the manual pulse generator is superposed on the travel distance by automatic operation in the direction of the selected handle feed axis on the coordinate (program coordinate) system after 3-dimensional coordinate conversion.

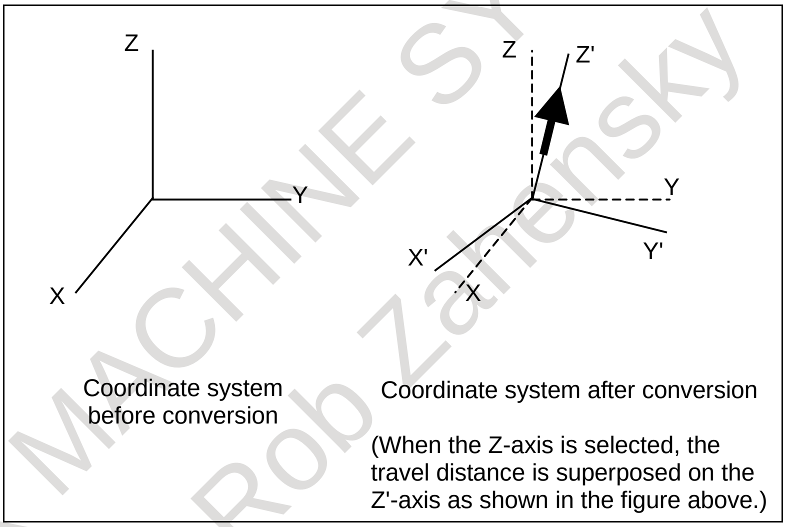


Fig. 3.3.1 Manual interruption of 3-dimensional coordinate system conversion

Explanation

- Interruption

This function performs interruption only when all of conditions 1 to 6 below are met. Otherwise, interruption does not occur.

1. During the automatic operation mode (MEM mode, MDI mode, or RMT mode)
2. During the 3-dimensional coordinate conversion mode
3. During the cutting mode (G01, G02, G03, or other G code for cutting is being executed.)
4. When interrupt is done to three axes of the target as 3-dimensional coordinate conversion
5. When 3-dimensional coordinate conversion manual interrupt enable/disable switch signal NOT3DM is set to 0
6. When the tool axis direction, the tool axis right-angle direction, or the tool tip center rotation is not selected for the 3-dimensional manual feed (when input signals ALNGH, RGHTH, and RNDH are all "0")

The feedrate superposed along the 3-dimensional coordinate conversion mode does not exceed the maximum cutting speed of each axis.

- Manual interruption and coordinate system

When this function operates, the absolute coordinate values do not include the travel distance by manual interruption. The absolute coordinate values are therefore not updated even when the manual pulse generator is rotated.
The machine coordinate values and relative coordinate values include the travel distance by manual interruption. However, the values are represented on the coordinate system before being converted. As shown in Fig. 3.3.1, the travel distance of the manual handle interruption is superposed along the Z'-axis. The travel distance is displayed using the X, Y, and Z coordinates of the coordinate system before being converted.
The travel distance by this function can be monitored on the manual handle interruption screen. Bit 3 (DMK) of parameter No. 5402 specifies whether the coordinate system (X, Y, Z) before being converted or the coordinate system (X', Y', Z') or after being converted is used to display the amount of interruption by the manual handle.

Signals

3-dimensional coordinate system conversion manual interrupt enable/disable switch signal NOT3DM<Gn347.7>

[Classification] Input signal
[Function] Enables or disables the 3-dimensional coordinate conversion manual interrupt function.
[Operation] When this signal is set to “0”, the 3-dimensional coordinate conversion manual interrupt function is enabled.
When this signal is set to “1”, the 3-dimensional coordinate conversion manual interrupt function is disabled.

3-dimensional coordinate conversion manual interrupt mode in-progress signal D3MI<Fn347.7>

[Classification] Output signal
[Function] Reports the state of the 3-dimensional coordinate conversion manual interrupt.
[Operation] The signal is set to “1” when all of the following conditions are met.
- During the automatic operation mode (MEM mode, MDI mode, or RMT mode)
- During 3-dimensional coordinate system conversion mode
- During cutting mode (G01, G02, G03, or other G code for cutting is being executed.)
- When 3-dimensional coordinate conversion manual interrupt enable/disable switch signal NOT3DM is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn347	NOT3DM							

	#7	#6	#5	#4	#3	#2	#1	#0
Fn347	D3MI							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5402					DMK			

[Input type] Parameter input
[Data type] Bit path

#3 DMK The manual handle interruption screen is displayed:
0: Using the program coordinate system (post-conversion coordinate system).
1: Using the workpiece coordinate system (pre-conversion coordinate system).
This parameter is valid only in 3-dimensional coordinate conversion mode.

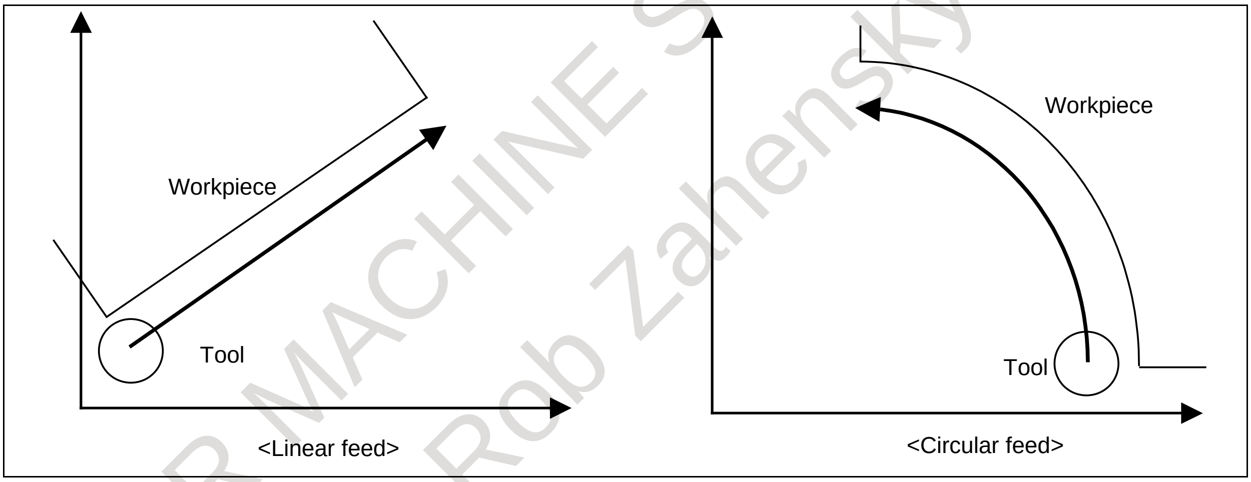
Note
Handle interruption is disabled during execution of a G68 or G69 block.

3.4 MANUAL LINEAR/CIRCULAR INTERPOLATION

Overview
In manual handle feed or jog feed, the following types of feed operations are possible along with the conventional feed operation with simultaneous single-axis control (for X, Y, Z, or other axis).

- Feed along a tilted straight line in the XY, YZ, or ZX plane based on simultaneous 2-axis control (linear feed)
- Feed along a circle in the XY, YZ, or ZX plane based on simultaneous 2-axis control (circular feed)

When using this function, please enable "Manual handle feed (bit 0 (HPG) of parameter No.8131 is 1)". When using the fourth or fifth manual pulse generator, the function "Manual handle feed 4/5-unit" is needed.



NOTE
1 Two control axes should be included in the three standard axes.
2 In the following, only the case of the XY plane is described. If any other plane is used, replace XY with YZ or ZX. (Refer to the table below.)

XY plane	YZ plane	ZX plane
X	Y	Z
Y	Z	X

3 Either of two control axes should be reference axis (parameter No. 1031).
Example) Parameter setting of the machine which has X-axis, Y-axis and Z-axis
- When performing manual linear/circular interpolation on X-Y plane
- When performing manual linear/circular interpolation on Y-Z plane
- When performing manual linear/circular interpolation on Z-X plane

The setting of the reference axis (parameter No. 1031) influences functions other than this function, and set it carefully enough.

Explanations

- Line/circle definition

A line or circle definition is not required for axis direction feed. For linear or circular feed, however, a line or circle must be defined, using the interface described later. (For example, in the case of circular feed, the center and radius of the circle must be specified.)

- Interface area

Part of the R area in the PMC is used for line and circle definitions. Set the necessary data in this area using the PMC or macro executor. See "Data setting" below for information about what data needs to be set at each address.

- Data setting

This function uses R960 to R979 of PMC as input data and R980 to R989 as output data for line and circle definitions.
When bit 3 (MRI) of parameter No. 7106 is set to 1, an arbitrary R signal can be assigned to input data. Likewise, an arbitrary R signal can be assigned to output data by setting bit 4 (MRO) of parameter No. 7106 to 1. In the case of input data, the head address of the R signal used is set in parameter No. 13541. In the case of output data, the head address of the R signal used is set in parameter No. 13542. (Input data uses an area of 20 bytes, and output data uses an area of 10 bytes.)

(a) Input data (PMC -> CNC)

A straight line or circle is defined by setting the following data.
By default, add "960". When bit 3 (MRI) of parameter No. 7106 is set to 1, add the value set in parameter No. 13541.

<table>
<tr><th rowspan="2">Data address</th><th rowspan="2">Number of bytes</th><th colspan="2">Setting</th></tr>
<tr><th>Linear feed</th><th>Circular feed</th></tr>
<tr><td>+0</td><td>1</td><td colspan="2">(Reserve) Do not use.</td></tr>
<tr><td>+1</td><td>1</td><td colspan="2">Linear or circular feed selection
<table>
<tr><th>Set value</th><th>Description</th></tr>
<tr><td>0</td><td>Neither linear nor circular feed is performed.</td></tr>
<tr><td>1</td><td>Linear feed is carried out.</td></tr>
<tr><td>2</td><td>Clockwise circular feed is performed.</td></tr>
<tr><td>3</td><td>Counterclockwise circular feed is performed.</td></tr>
</table></td></tr>
<tr><td>+2 to +5</td><td>4</td><td>Approach direction (X-axis direction)</td><td>Center of the circle (X_o)</td></tr>
<tr><td>+6 to +9</td><td>4</td><td>Approach direction (Y-axis direction)</td><td>Center of the circle (Y_o)</td></tr>
<tr><td>+10 to +13</td><td>4</td><td>Distance (P) between the origin and a given line</td><td>Radius (R) of the circle</td></tr>
</table>

<table>
<tr><th rowspan="2">Data address</th><th rowspan="2">Number of bytes</th><th colspan="2">Setting</th></tr>
<tr><th>Linear feed</th><th>Circular feed</th></tr>
<tr><td>+14</td><td>1</td><td>Direction of cutting (direction of movement when the guidance handle is rotated forward)
<table>
<tr><th>Set value</th><th>Description</th></tr>
<tr><td>0</td><td>+90° with relation to the approach direction</td></tr>
<tr><td>1</td><td>-90° with relation to the approach direction</td></tr>
</table></td><td>Which side to be machined
<table>
<tr><th>Set value</th><th>Description</th></tr>
<tr><td>0</td><td>Inside of the circle</td></tr>
<tr><td>1</td><td>Outside of the circle</td></tr>
</table></td></tr>
<tr><td>+15</td><td>1</td><td colspan="2">Control flags
<table>
<tr><td>Bit 0 to bit 5</td><td>Must be set to 0.</td></tr>
<tr><td>Bit 6</td><td>Prohibited area when the limit function is enabled (Bit 7 is ”1”)
0: Area in the same direction as approach direction
1: Area in the opposite direction as approach direction</td></tr>
<tr><td>Bit 7</td><td>Whether the limit function is enabled or disabled
0: Disabled
1: Enabled</td></tr>
</table></td></tr>
<tr><td>+16</td><td>1</td><td colspan="2">Plane selection</td></tr>
<tr><td>+17 and +18</td><td>2</td><td colspan="2">(Reserve) Do not use. (Must be set to 0.)</td></tr>
<tr><td>+19</td><td>1</td><td colspan="2">Setting change notification</td></tr>
</table>

(b) Output data (CNC -> PMC)

By default, add "980". When bit 4 (MRO) of parameter No. 7106 is set to 1, add the value set in parameter No. 13542.

Data address	Number of bytes	Setting	
		Linear feed	Circular feed
+0 to +3	4	The distance from the current position to the given line is output.	The distance from the current position to the given circle is output.
+4 to +9	6	(Reserve) Do not use.	

NOTE

1 When the multi-PMC function is used, only the first PMC R address is available.

2 About the settings of parameters No. 13541 and No.13542

<1>Set a value that is a multiple of 4 (0, 4, 8, etc.).

<2>The addresses of input data and output data must not overlap.

<3>When a multi-path system is used, set a value that does not overlap the data address used in any other path.

<4>The range of the R address differs depending on the PMC used and the memory size. Check the specifications of the PMC, and set a value within the valid range.

If any setting other than the above items (<1> to <4>) is made, alarm PW5390 "R-ADDRESS SETTING IS ILLEGAL" is issued.

3 Hereinafter, the default data addresses (R960 to R979 for input data and R980 to R989 for output data) are used in the explanation of the setting data.

Setting for the plane selection (R976)
Set each plane as follows.

R976	#7	#6	#5	#4	#3	#2	#1	#0	Hexadecimal number
XY plane	0	0	1	0	0	0	0	1	21H
YZ plane	0	0	1	1	0	0	1	0	32H
ZX plane	0	0	0	1	0	0	1	1	13H

Setting for setting change notification (R979)
The definition of a straight line or circle (data of R961 to R975) can be set or changed in the manual operation mode (manual handle feed mode or jog feed mode). This data notifies the CNC that the definition has been changed.
After setting the data of R961 to R975, set 0 in the setting change notification (R979). When the value of R979 is changed to 0, the CNC judges that the data has been changed. After reading the data of R961 to R975, the CNC sets 1 in the setting change notification (R979). Afterwards, the linear feed or the circular feed is executed based on the read data until the setting change notification (R979) is changed to 0 again.

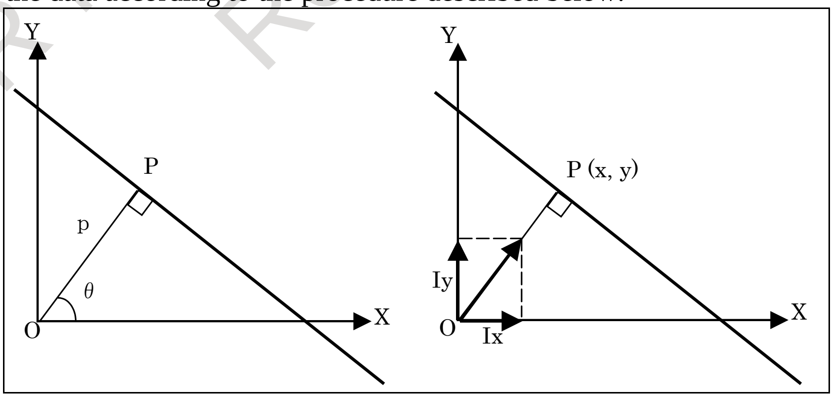
NOTE
1. When setting the setting change notification (R979), make sure that each axis is stopped.
2. While the setting change notification (R979) is set to "0", do not change the data.

- Setting for linear feed

Assuming that p is the length of a perpendicular line between the origin (O) and the straight line, that P(x,y) is the intersection of the straight line and the perpendicular line, and that θ is the angle formed by the perpendicular line and the positive X-axis, the straight line can be defined as:
$x \cdot \cos\theta + y \cdot \sin\theta = p$

NOTE
The "origin" refers to that in the absolute coordinate system. Coordinates X and Y used in this description represent the center of the tool.

Based on this, set the data according to the procedure described below.



Setting data

<1>	R961	:	Linear feed
<2>	R962 - R965	:	Approach direction Ix · 2^{30}
<3>	R966 - R969	:	Approach direction Iy · 2^{30}
<4>	R970 - R973	:	Distance between origin and straight line p
<5>	R974	:	Direction in which the guidance handle moves
<6>	R975	:	Whether the limit function is enabled or disabled
<7>	R979	:	Setting change notification

<1> Select linear feed. (R961)
Set R961 to "1".

<2> <3> Specify the approach direction. (R962 to R969)
Specify the X and Y components (Ix, Iy) of a unit vector (±cosθ, ±sinθ), which is parallel to perpendicular line OP, each with four bytes. Set a value multiplied by 2^{30}.
R962 to R965 = $Ix \cdot 2^{30}$
R966 to R969 = $Iy \cdot 2^{30}$
The tool moves in the direction indicated with this unit vector when the approach handle is rotated in the forward direction. The unit of data is the least input increment set for the reference axis by parameter No. 1031. (For example, when metric input is used for IS-B, the unit is 0.001 mm.)

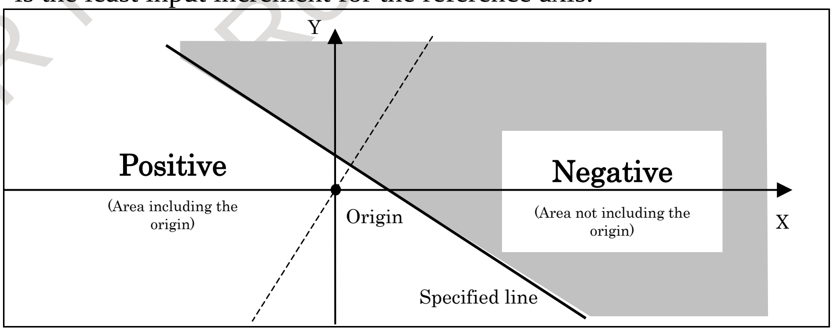
<4> Specify length P of the perpendicular line from the origin to the straight line with four bytes. (R970 to R973)
Length P must satisfy the following equation:

$P = \pm| \overrightarrow{OP} |$

According to the side of the specified straight line on which the approach is started, the sign changes as follows.
(Refer to the following figure.)

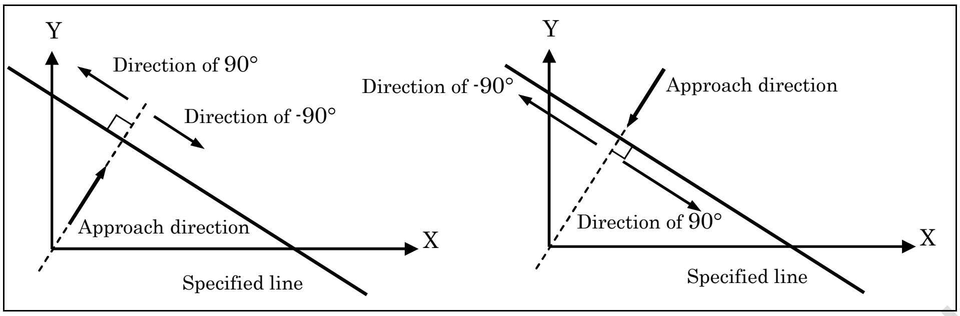
Area including the origin : Positive
Area not including the origin : Negative

(Set a "negative" value when the tool is approached from the direction opposite to the direction set in <2> and <3>.)
The unit of P is the least input increment for the reference axis.



<5> Specify the cutting direction. (R974)
Specify the direction of travel for the forward rotation of the guidance handle, using R974.
The meaning of the setting is as follows:

0: Direction towards +90° from the approach direction
1: Direction towards -90° from the approach direction



<6> Specify whether to enable the limit function. (Bit 7 of R975)

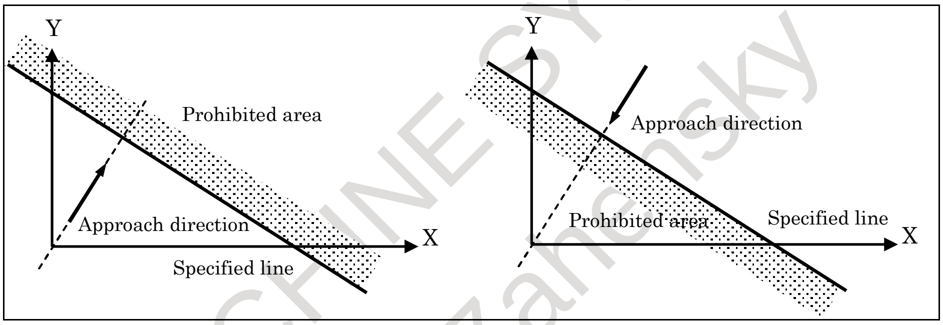
To disable the limit function, set bit 7 of R975 to "0".

To enable the limit function, set bit 7 of R975 to "1".

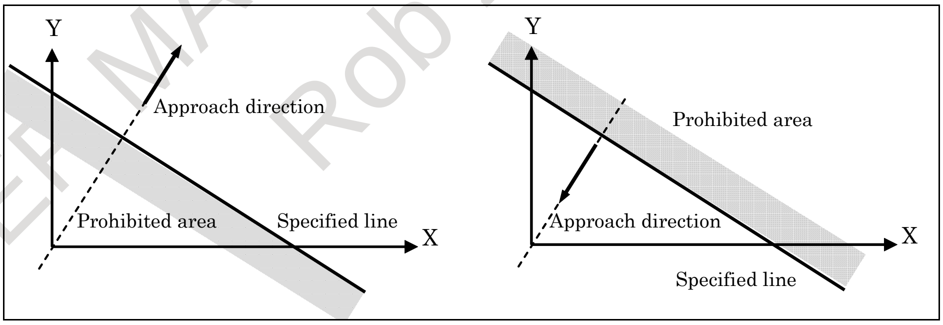
When the limit function is enabled, an area is set where the tool is not allowed to enter. The area is delimited with a specified line. When you attempt to bring the tool into the prohibited area, using manual handle feed or jog feed, the tool decelerates and stops. Operation on the specified line is possible.

The prohibited area is set as shown below, according to the setting of the approach direction (R962 to R965, R966 to R969) and the setting of the prohibited area (Bit 6 of R975).

- When bit 6 of R975 is set to "0"



- When bit 6 of R975 is set to "1"



<7> Set the setting change notification. (R979)

Set R979 to "0".

This completes the straight line definition.

During linear feed, the distance to the specified line is calculated, using the following equation, and output to R980 to R983 (4 bytes).

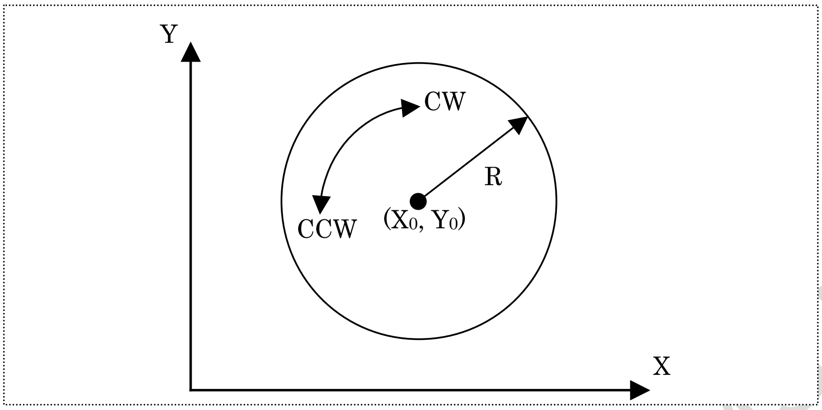
The unit of the data is the least input increment for the reference axis.

$F(X,Y) = p - (Ix \cdot X + Iy \cdot Y)$

(Where X, Y = Current X- and Y-axis positions)

- Setting for circular feed

Specify the data according to the procedure below.



Setting data

<1> R961 : Circular feed and direction of rotation (CW/CCW)
<2> R962 to R965 : Center of the circle X_0
<3> R966 to R969 : Center of the circle Y_0
<4> R970 to R973 : Radius R
<5> R974 : Which side to be machined (inside or outside)
<6> R975 : Whether the limit function is enabled or disabled
<7> R979 : Setting change notification

<1> Specify circular feed and the direction of rotation. (R961)
Set R961 to "2" or "3".
If R961 is set to "2", the tool moves along the circle clockwise, when the guidance handle is rotated in the forward direction.
If R961 is "3", the tool moves along the circle counterclockwise, when the guidance handle is rotated in the forward direction.

<2> <3> Specify the coordinates (X_0, Y_0) of the center of the circle. (R962 to R965, R966 to R969)

NOTE
X and Y used in this description refer to coordinates in the absolute coordinate system.

R962 to R965 = X_0
R966 to R969 = Y_0
Each coordinate is four bytes. The unit of the data is the least input increment for the reference axis.

<4> Specify radius R. (R970 to R973)
R970 to R973 = R
The radius R is four bytes. The unit of the data is the least input increment for the reference axis.

<5> Specify which side is to be machined, the inside or outside of the circle. (R974)
Set R974 to "0" or "1".
If R974 is 0, the inside of the circle is machined.
If R974 is 1, the outside of the circle is machined.
When the approach handle is rotated, the tool moves along a straight line normal to the specified circle. The direction of the tool movement is determined according to the setting of R974.

When the approach handle is rotated in the forward direction, the direction of the tool movement (approach direction) is as follows:

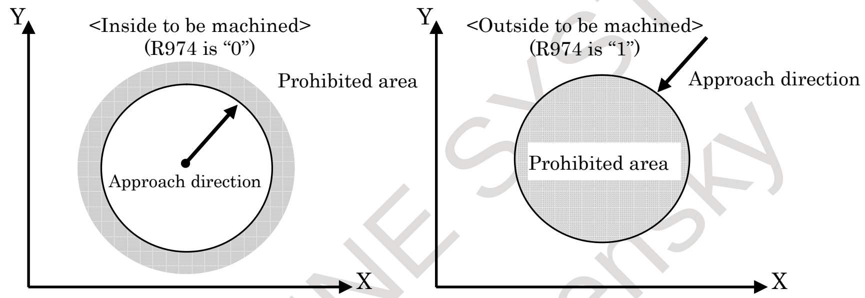
- When the inside of the circle is to be machined (R974 = “0”), the tool moves from the center of the circle to the circumference.
- When the outside of the circle is to be machined (R974 = “1”), the tool moves toward the center of the circle.

<6> Specify whether to enable or disable the limit function. (Bit 7 of R975)

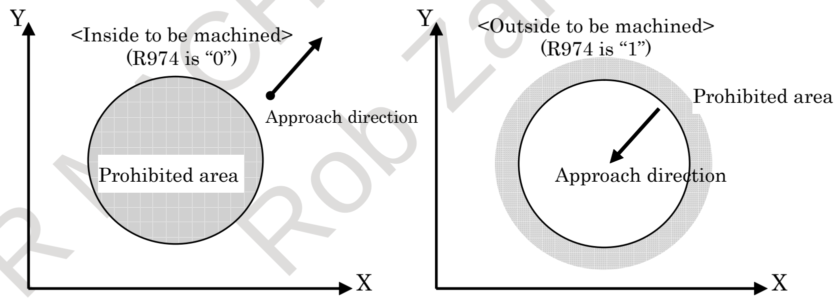
To disable the limit function, set bit 7 of R975 to 0.
To enable the limit function, set bit 7 of R975 to 1.

When the limit function is enabled, an area is set where the tool is not allowed to enter. The area is either inside or outside of the specified circle. When you attempt to bring the tool into the prohibited area, using manual handle feed or jog feed, the tool decelerates and stops. Operation on the specified circle is possible.
Whether the inside or outside of the circle is set as the prohibited area is determined according to the setting of R974 that specifies which side (inside or outside of the circle) is to be machined and the setting of the prohibited area (Bit 6 of R975).

- When bit 6 of R975 is set to "0"



- When bit 6 of R975 is set to "1"



<7> Set the setting change notification. (R979)

Set R979 to "0".

This completes the circle definition.

During circular feed, the distance to the specified circle is calculated, using the following equation, and output to R980 to R983 (4 bytes). The unit of the data is the least input increment for the reference axis.
When the inside of the circle is to be machined:

$$F(X,Y) = R - \sqrt{(X - X_0)^2 + (Y - Y_0)^2}$$

When the outside of the circle is to be machined:

$$F(X,Y) = \sqrt{(X - X_0)^2 + (Y - Y_0)^2} - R$$

- When neither linear nor circular feed is carried out

Specify the following data.

Setting data

<1> R961	: 0 (Neither linear nor circular feed is carried out)
<2> R962 to R965	: (Need not be specified)
<3> R966 to R969	: (Need not be specified)
<4> R970 to R973	: (Need not be specified)
<5> R974	: (Need not be specified)
<6> R975	: (Need not be specified)
<7> R976	: (Need not be specified)
<8> R979	: Setting change notification

<1> Linear or circular feed selection
Reset R961 to "0".
If R961 is set to 0, both the guidance and approach handles become ineffective.
The tool will not move, even if these handles are rotated.

<2> to <7> It is unnecessary to set R962 to R976.

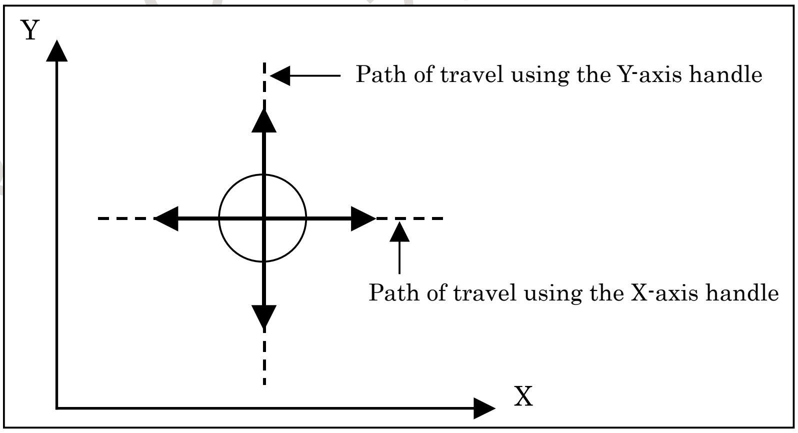
<8> Set the setting change notification. (R979)
Set R979 to "0".

The values of R980 to R983 (distance to a given line or circle) are output as "0".

- Manual handle feed

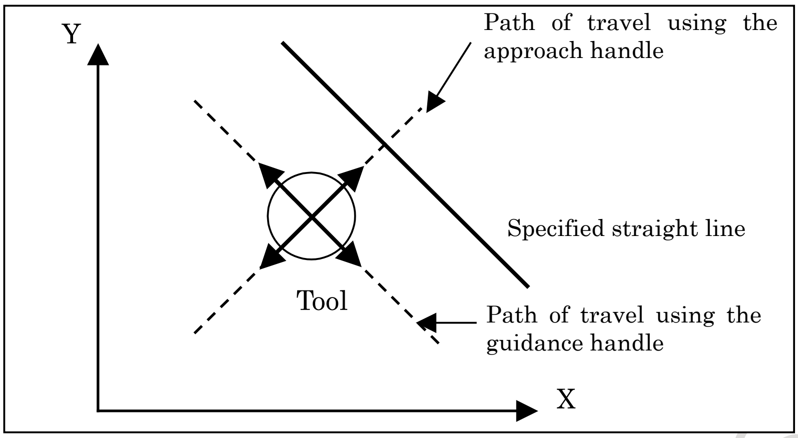
In manual handle feed, the tool can be moved along a specified axis (X-axis, Y-axis, Z-axis, or N^{th}-axis), along a rotated straight line (linear feed), or along a circle (circular feed).

(1) Feed along a specified axis (simultaneous 1-axis control)
A single manual handle operation can move the tool along the desired axis, such as X-axis, Y-axis, or Z-axis, on a simultaneous 1-axis control basis.



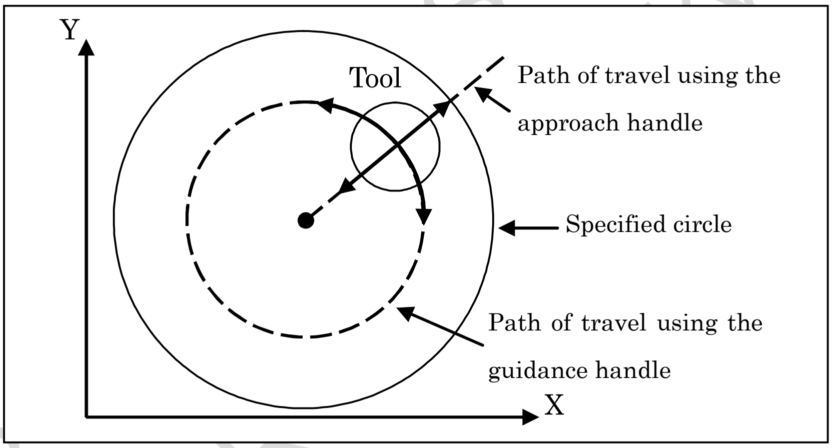
Feed along a specified axis

(2) Linear feed (simultaneous 2-axis control)
A single manual handle operation can move the tool in parallel to the straight line on a simultaneous 2-axis control basis. This manual handle is referred to as the "guidance handle". Also, the tool can be moved at right angles to a specified straight line on a simultaneous 2-axis control basis. This manual handle is referred to as the "approach handle". When the guidance handle or approach handle is turned clockwise or counterclockwise, the tool travels forward or backward along the respective path.



Linear feed

(3) Circular feed (simultaneous 2-axis control)
A single manual handle operation can move the tool from the current position along a concentric circle that has the same center as a specified circle on a simultaneous 2-axis control basis. This manual handle is referred to as the guidance handle. Also, a single manual handle operation can move the tool along a straight line normal to a specified circle on simultaneous 2-axis control. This manual handle is referred to as the approach handle. When the guidance handle or approach handle is turned clockwise or counterclockwise, the tool travels forward or backward along the respective path.



Circular feed

- Feedrate for manual handle feed

The feedrate depends on the speed at which a manual handle is turned. The distance to be traveled by the tool (along a tangent in the case of linear or circular feed) per manual handle pulse can be selected using the manual handle feed travel distance magnification signal MP1<Gn019.4> or MP2<Gn019.5>.
The approach handle clamp feedrate can be set using parameter No. 7160. The guidance handle clamp feedrate can be set using parameter No. 7161. Set the parameter No. 7117 to 0.

- Manual handle selection

Five manual pulse generator interfaces are provided to allow up to five manual handles to be connected. The mode of the manual handles connected to the interfaces (whether to use the manual handle as a handle for feed along an axis, as a guidance handle, or as an approach handle) can be selected by the manual linear/circular interpolation signal MHLCn<Gn544.0 to Gn544.4> and manual handle usage selection signal MHUSn<Gn545.0 to Gn545.4>, which are provided for each interface. By switching these signals, a single manual handle can be used for multiple purposes.

- Direction of movement using manual handles

When the tool is moved along a straight line or circle using the guidance handle or approach handle, the direction of the tool movement (for example, whether to make a clockwise or counterclockwise movement along a circle) when the handle is turned clockwise or counterclockwise can be specified using the interfaces described above.

- Jog feed

In jog feed, the tool can be moved along a specified axis (X-axis, Y-axis, Z-axis, etc.), along a rotated straight line (linear feed), or along a circle (circular feed).

(1) Feed along a specified axis (simultaneous 1-axis control)

While a feed axis and its direction are specified with the feed axis direction selection signals +J1~+J8 <Gn100> and -J1~-J8 <Gn102>, the tool moves in the specified axis direction.

When feeding the axis of selected plane (R976) of manual linear/circular interpolation in each axis direction, feedrate is the value of parameter No.1410 (parameter No.1415 is used when bit 4 (JRV) of parameter No.1402 is set to 1). The feedrate can be overridden (0.01% to 655.34%) using the manual feedrate override signal *JV0 to *JV15 <Gn010,Gn011>.

NOTE

1. Two axes of the selected plane cannot be fed at the same time in each axis direction. (When feed axis direction selection signals of two axes are set to "1" simultaneously, the two axes decelerate and stop.)
2. The manual rapid traverse selection signal RT <Gn019.7> is invalid for two axes of selected plane. (Even if RT is set to "1", a rapid traverse rate is not selected.)

When the axis excluding the plane axis selected by R976 of manual linear/circular interpolation is fed in each axis direction, the feederate is the value of parameter No. 1423. The feedrate can be overridden (0.01% to 655.34%) using the manual feedrate override signal *JV0~*JV15 <Gn010, Gn011>.

(2) Linear feed (simultaneous 2-axis control)

By defining a straight line, the tool can be moved as follows:

- While a feed axis and its direction are selected using the feed axis direction selection signal +Jg<Gn086.0> or –Jg<Gn086.1>, the tool moves along a straight line parallel to the specified straight line on a simultaneous 2-axis control basis.
- While a feed axis and its direction are selected using the feed axis direction selection signal +Ja<Gn086.2> or -Ja<Gn086.3>, the tool moves along a straight line perpendicular to the specified straight line on a simultaneous 2-axis control basis.

The feedrate in the tangential direction is specified in parameter No. 1410 (Parameter No.1415 is used when bit 4 (JRV) of parameter No. 1402 is set to 1). The feedrate can be overridden (0.01% to 655.34%) using the manual feedrate override signal *JV0 to *JV15 <Gn010, Gn011>. The manual rapid traverse selection signal RT<Gn019.7> is invalid. (Even if RT is set to "1", a rapid traverse rate is not selected.)

(3) Circular feed (simultaneous 2-axis control)

By defining a circle, the tool can be moved as follows:

- While a feed axis and its direction are selected using the feed axis direction selection signal +Jg<Gn086.0> or –Jg<Gn086.1>, the tool moves from the current position along the concentric circle that has the same center as the specified circle.
- While a feed axis and its direction are selected using the feed axis direction selection signal +Ja<Gn086.2> or -Ja<Gn086.3>, the tool moves along a straight line normal to the specified circle.

The feedrate in the tangential direction is specified in parameter No. 1410 (Parameter No.1415 is used when bit 4 (JRV) of parameter No. 1402 is set to 1). The feedrate can be overridden (0.01% to 655.34%) using the manual feedrate override signal *JV0 to *JV15<Gn010, Gn011>. The manual rapid traverse selection signal RT<Gn019.7> is invalid. (Even if RT is set to "1", a rapid traverse rate is not selected.)

- Manual handle feed in JOG mode

Even in JOG mode, manual handle feed can be enabled using bit 0 (JHD) of parameter No. 7100. In this case, however, manual handle feed is enabled only when the tool is not moved using jog feed.

Basic procedure

(1) Select the manual operation mode.
To perform manual handle feed, select manual handle feed mode. To perform jog feed, select jog feed mode.

(2) Define a line or circle.
See the "Line/circle definition" in the Explanations section.

(3) Move the tool using manual handle feed or jog feed.

To perform the manual handle feed

First, select either single-axis feed along the X-, Y- or Z-axis, or simultaneous two-axis feed along a specified line or circle in the specified plane, using manual linear/circular interpolation signal MHLCn.

To execute single-axis feed along the X-, Y- or Z-axis, select the axis along which the tool will move when the manual handle is turned, using the manual handle feed axis selection signals HSnA, HSnB, HSnC, HSnD, and HSnE.

To execute simultaneous two-axis feed in the specified plane, select either approach handle or guidance handle, using the manual handle usage selection signal (MHUSn). By selecting the plane, the two axes along which the tool will move when the manual handle is turned are selected.

Turning the manual handle will move the tool along the selected axis. The feedrate varies with the speed at which the manual handle is turned.

The amount the tool will be moved when the manual handle is turned by one pulse can be specified using manual handle feed amount selection signals MP1 and MP2.

To perform the jog feed

Select the feed axis and the direction in which the tool is to be moved, using the feed axis and direction selection signals +Jg, -Jg, +Ja, -Ja. While the feed axis is specified, the tool is moved along the specified axis or specified line or circle at the parameter-set feedrate (jog feedrate).

The manual feedrate override signals *JV0 to *JV15 can be used to apply an override to the feedrate. The manual rapid traverse selection signal RT is invalid.

To stop the feed of manual linear/circular interpolation

When jog feed is performed without carrying out the feed of manual linear/circular interpolation, specify the following data.

- Specify the R area in the PMC which is describe in the above-mentioned "When neither linear nor circular feed is carried out"
- Set Feed axis and direction selection signal +Jg, -Jg, +Ja, -Ja to “0”
- Set Manual linear/circular interpolation signal MHLC1 to MHLC5 to “0”

Limitations

- Mirror image

The mirror image function is not available during the manual operation. (Operate when the mirror image signal MI1 to MI3<Fn108> is set to "0" and the mirror image setting is off.)

> **NOTE**
> If the tool is operated with the mirror image function activated, the normal operation will not be guaranteed.

- Rotation direction

When this function is effective, bit 0 (HNGx) of parameter No. 7102 is invalid which is used to change the axis movement direction with respect to the rotation direction of the manual pulse generator. (The rotation direction of the manual pulse generator is always the same as the axis movement direction; that is, the operation is the same as when bit 0 (HNGx) of parameter No. 7102 is set to 0.)

- The amount of the shift for manual handle feed

When this function is effective, parameters No. 12350 and No. 12351 used to determine the magnification of manual handle feed for each axis are invalid and the values of parameter No. 7113 and No. 7114 are used. (Parameters No. 7131 and No. 7132 are used for the second manual pulse generator, parameters No. 7133 and No. 7134 for the third, parameters No. 7135 and No. 7136 for the fourth, and parameters No. 7137 and No. 7138 for fifth.)

- Incremental feed

This function is invalid for the incremental feed.

- Synchronous/composite control

If the axis is set as the synchronous/composite control axis, this function is invalid.

Note

> **NOTE**
> When using this function, please enable "Manual handle feed (bit 0 (HPG) of parameter No.8131 is 1)". When using the fourth or fifth manual pulse generator, the function "Manual handle feed 4/5-unit" is needed.

Signal

The following signals are related to jog feed and manual handle feed.

Feed axis and direction selection signal +Jg, -Jg, +Ja, -Ja <Gn086.0 to Gn086.3>

[Classification] Input signal

[Function] Selects a desired feed axis and direction in jog feed. The sign (+ or -) in the signal name indicates the feed direction. "g" and "a" after J indicates Guidance JOG and Approach JOG, respectively.

[Operation] For operation, these signals are the same as feed axis and direction selection signal +J1 to +J8, -J1 to –J8 <Gn100, Gn102>.
Refer to section "JOG FEED/ INCREMENTAL FEED".

Manual linear/circular interpolation signal MHLC1 to MHLC5 <Gn544.0 to Gn544.4>

[Classification] Input signal

[Function] Each of these signals selects the manual linear/circular interpolation function for the n-th manual pulse generator.

[Operation]
0: Manual linear/circular interpolation is disabled. The manual handle is used for axis direction feed.
1: Manual linear/circular interpolation is enabled. The manual handle is used for the purpose selected by the usage selection of manual linear/circular interpolation signal (MHUSn).

Usage selection of manual linear/circular interpolation signal MHUS1 to MHUS5 <Gn545.0 to Gn545.4>

[Classification] Input signal

[Function] Selects the guidance handle or approach handle for the n-th manual pulse generator when the manual linear/circular interpolation signal MHLCn is set to "1" for the manual linear/circular interpolation function.

[Operation] 0: The manual handle is used as the approach handle.
1: The manual handle is used as the guidance handle.

Selection of usage of the nth manual handle

MHLCn	MHUSn	Usage
0	-	Axis direction feed
1	0	Approach handle
1	1	Guidance handle

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn086					-Ja	+Ja	-Jg	+Jg
Gn544				MHLC5	MHLC4	MHLC3	MHLC2	MHLC1
Gn545				MHUS5	MHUS4	MHUS3	MHUS2	MHUS1

Alarm and message

Number	Message	Description
PW5390	R-ADDRESS SETTING IS ILLEGAL	Either the R address range of the PMC set by parameters No. 13541 and No. 13542 or the first address of the range is invalid.

Parameter

1031	Reference axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

The unit of some parameters common to all axes such as those for dry run feedrate and single-digit F1 feedrate may vary according to the increment system. An increment system can be selected by a parameter on an axis-by-axis basis. So, the unit of those parameters is to match the increment system of a reference axis. Set which axis to use as a reference axis.

Among the basic three axes, the axis with the finest increment system is generally selected as a reference axis.

> **NOTE**
> When this parameter is set to 0, the reference axis is 1'st axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1402				JRV				

[Input type] Parameter input
[Data type] Bit path

#4 JRV Jog feed or incremental feed is
0: Performed at feed per minute.

1: Performed at feed per rotation.

NOTE
1 Specify a feedrate in parameter No. 1423.
2 For the machining center system, the threading/synchronous feed is required.

1410	Dry run rate

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the dry run rate at the 100% position on the jog feedrate specification dial. The unit of data depends on the increment system of the reference axis. Setting this parameter to 0 results in alarm PS5009, "PARAMETER ZERO (DRY RUN)", being issued.

1415	Manual synchronous feedrate for manual linear/circular interpolation continuous feed at override 100%

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

NOTE
This parameter is valid only when bit 4 (JRV) of parameter No. 1402 is 1 (manual per revolution feed is enabled). If manual per revolution feed is disabled, the manual linear/circular interpolation continuous feedrate follows the dry run rate (parameter No. 1410). For a machining center system, the manual linear/circular interpolation continuous feedrate follows the dry run rate (parameter No. 1410) (feed per minute) even though bit 4 (JRV) of parameter No. 1402 is set to 1 when the threading/synchronous feed function is not specified.

1423	Jog feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
(1) When bit 4 (JRV) of parameter No. 1402 is set to 0, set jog feedrate (feed per minute) for each axis when manual feedrate override is 100%.
(2) When bit 4 (JRV) of parameter No. 1402 is set to 1, set jog feedrate (feed per revolution) for each axis when manual feedrate override is 100%.

> **NOTE**
> This parameter is clamped to the axis-by-axis manual rapid traverse rate (parameter No. 1424).

	#7	#6	#5	#4	#3	#2	#1	#0
7001	MFM							

[Input type] Parameter input
[Data type] Bit path

#7 MFM For the manual linear/circular interpolation function, modifying a value specified with a command during jog feed in the guidance direction (approach direction):
0: Immediately starts moving according to the new value.
1: Stops moving.

	#7	#6	#5	#4	#3	#2	#1	#0
7100							THD	JHD

[Input type] Parameter input
[Data type] Bit path

#0 JHD Manual handle feed in JOG feed mode or incremental feed in the manual handle feed is:
0: Invalid.
1: Valid.

#1 THD In the TEACH IN JOG mode, the manual pulse generator is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
7106				MRO	MRI			

[Input type] Parameter input
[Data type] Bit path

#3 MRI Internal relay (the R signal) of PMC that uses it with input data in manual linear/circular interpolation:
0: R960 to R979 are used.
1: The address set by parameter No.13541 is used as the start address.

> ⚠ **WARNING**
> Set an appropriate value to parameter No.13541 beforehand when you set 1 to this parameter.
> If the set address in parameter No.13541 is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used internal relay.

#4 MRO Internal relay (the R signal) of PMC that uses it with output data in manual linear/circular interpolation:
0: R980 to R989 are used.
1: The address set by parameter No.13542 is used as the start address.

> ⚠ **WARNING**
> Set an appropriate value to parameter No.13542 beforehand when you set 1 to this parameter.
> If the set address in parameter No.13542 is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used internal relay.

7113	Manual handle feed magnification m

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 2000

This parameter sets the magnification m when manual handle feed movement selection signals MP1 and MP2 are set to "0" and "1".

7114	Manual handle feed magnification n

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 2000

This parameter sets the magnification when manual handle feed movement selection signals MP1 and MP2 are set to "1".

7131	Manual handle feed magnification m2 / 2nd. manual pulse generator
7132	Manual handle feed magnification n2 / 2nd. manual pulse generator
7133	Manual handle feed magnification m3 / 3rd. manual pulse generator
7134	Manual handle feed magnification n3 / 3rd. manual pulse generator
7135	Manual handle feed magnification m4 / 4th. manual pulse generator
7136	Manual handle feed magnification n4 / 4th. manual pulse generator
7137	Manual handle feed magnification m5 / 5th. manual pulse generator
7138	Manual handle feed magnification n5 / 5th. manual pulse generator

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 2000

The 'mx' sets magnification when manual handle feed amount selection signals MPx1 = 0, MPx2 = 1. The 'nx' sets magnification when manual handle feed amount selection signals MPx1 = 1, MPx2 = 1.

7160	Approach handle clamp feedrate

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis。
[Valid data range] Refer to standard parameter setting table (C)

Approach handle clamp feedrate is set.

7161	Guidance handle clamp feedrate

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis。
[Valid data range] Refer to standard parameter setting table (C)

Guidance handle clamp feedrate is set.

13541	The head address of the R signal used by the input data in the manual linear/circular interpolation

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 59999

The head address of the internal relay(R signal) of PMC used by the input data in the manual linear/circular interpolation is set. 20-byte area is required for input data from the set address.

NOTE

1 When this parameter is set, the power must be turned off before operation is continued.
2 This parameter is valid when bit 3 (MRI) of parameter No.7106 is set to 1.
3 About setting parameters Nos. 13541 and 13542
   (1) Set the value which becomes the multiple of four. (0, 4, 8, …)
   (2) The address of output data and input data must not be duplicated.
   (3) When the multi-path system is used, set the value which does not duplicate the data address used in other path systems.
   (4) The range in R address is different depending on PMC used and the memory. Confirm the specification of PMC, and set the value within the range which can be used. (Example: First PMC memory B -> R0 to R7999)

   When the settings other than the above-mentioned ((1) to (4)) were done, the alarm PW5390, "R-ADDRESS SETTING IS ILLEGAL" is issued.
4 When this parameter is 0, an internal relay from address R0 is used.

⚠ WARNING

If the set address in this parameter is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used internal relay.

13542	Head address of the R signal used by the output data in the manual linear/circular interpolation

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 59999

The head address of the internal relay(R signal) of PMC used by the output data in the manual linear/circular interpolation is set. 10-byte area is required for output data from the set address.

NOTE

1 When this parameter is set, the power must be turned off before operation is continued.
2 This parameter is valid when bit 4 (MRO) of parameter No.7106 is set to 1.
3 About setting parameters Nos. 13541 and 13542
 (1) Set the value which becomes the multiple of four. (0, 4, 8, …)
 (2) The address of output data and input data must not be duplicated.
 (3) When the multi-path system is used, set the value which does not duplicate the data address used in other path systems.
 (4) The range in R address is different depending on PMC used and the memory. Confirm the specification of PMC, and set the value within the range which can be used. (Example: First PMC memory B -> R0 to R7999)
 When the settings other than the above-mentioned ((1) to (4)) were done, the alarm PW5390, “R-ADDRESS SETTING IS ILLEGAL” is issued.
4 When this parameter is 0, an internal relay from address R0 is used.

⚠ **WARNING**

If the set address in this parameter is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used internal relay.

Standard parameter setting table (C) velocity and angular velocity parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-A	0.01	0.00 to +999000.00
	IS-B	0.001	0.000 to +999000.000
	IS-C	0.0001	0.0000 to +99999.9999
inch/min	IS-A	0.001	0.000 to +96000.000
	IS-B	0.0001	0.0000 to +9600.0000
	IS-C	0.00001	0.00000 to +4000.00000

3.5 HANDLE-SYNCHRONOUS FEED

Overview

The tool is fed at a program-specified feedrate or at a feedrate that matches a dry run feedrate in cutting feed blocks (such as linear interpolation (G01) and circular interpolation (G02 and G03)) during automatic operation. This function enables the tool to be fed in synchronization with the rotation of a manual handle (manual pulse generator).

The manual linear/circular interpolation signals MHLCn <Gn544.0-4> and the usage selection of manual linear/circular interpolation signals MHUSn <Gn545.0-4> select the manual handle with which the tool is to be synchronized.

The feedrate for handle-synchronous feed is controlled in such a way that the tangential feedrate is commensurate with the rotation speed of the manual handle.

The travel distance of the tool per manual handle pulse (travel distance amount of tangential movement) is determined by the manual handle feed travel distance selection signals MP1 <Gn019.4> and MP2 <Gn019.5>.

The combination of signals determines which feedrate (program-specified feedrate (F command), a dry run feedrate, or a feedrate synchronized with the rotation of the manual handle) is to be used in a cutting feed block, as shown in the table below.
These signals can be switched even in the middle of a block.

Dry run signal DRN	Handle-synchronous feed signal HREV	Cutting feedrate
0	0	Program-specified feedrate
0	1	Feedrate synchronized with the rotation of the manual handle
1	0	Dry run feedrate
1	1	Dry run feedrate

It is possible to make effective only one direction of rotation of a manual handle. When bit 5 (HSR) of parameter No. 7106 is set to 1, the effective direction of the manual pulse generator is only CW (clockwise) if the handle rotation direction selection signal HDSR <Gn193.3> is set to 0. Also, the effective direction of the manual pulse generator is only CCW (counterclockwise) if the handle rotation direction selection signal HDSR <Gn193.3> is set to 1.
When bit 5 (HSR) of parameter No. 7106 is set to 0, both directions of the manual pulse generator are effective as usual.

Limitations

- Unit system of the control axis

The unit system of the control axis is decided by the setting for the reference axis selected by parameter No. 1031.

- Manual mode

This function cannot be used in manual mode.

- Different magnification in each axis

Manual handle feed axis is not applied when this function is executing, it is not possible to apply a different magnification to the manual handle pulse on a per-axis basis.

Note

NOTE

1 The direction of manual handle rotation does not influence the direction of tool movement. That is to say, rotating the manual handle backward does not cause the tool to reverse.
Handle-synchronous feed disregards the sign of the pulse from the manual pulse generator. (The absolute values of pulses are used.) Therefore, the tool moves along a programmed path through a distance that matches the rotation number of the manual handle, regardless of the direction of rotation.

2 If the maximum feedrate of the handle is set to the setting of parameter No. 1434 (maximum manual handle feedrate switch signal HNDLF <Gn023.3> = “1”), the maximum feedrate for the basic axes is effective.
The setting of parameter No. 1434 for any other axis is not effective.

3 It is not possible to make the maximum feedrate effective on an axis-by-axis basis.

4 When using this function, enable “Manual linear/circular interpolation” and "Manual handle feed (bit 0 (HPG) of parameter No.8131 is 1)". When using the fourth or fifth manual pulse generator, the function "Manual handle feed 4/5-units" is needed.

Signal

Manual handle feed amount selection signals (Incremental feed signals) MP1, MP2 <Gn019.4, Gn019.5>

[Classification] Input signal

[Function] This signal selects the distance traveled per pulse from the manual pulse generator during the manual handle feed or manual handle interrupt. It is also used for incremental feed.
The following table shows the correspondence between the signals and travel distances.

Manual handle feed amount selection signals		Distance traveled
MP2	MP1	Manual handle feed
0	0	Least input increment x 1
0	1	Least input increment x 10
1	0	Least input increment x m *1
1	1	Least input increment x n *1

*1 Magnifications m and n are specified using parameters No. 7113 and 7114.

> ⚠ **CAUTION**
> 1 Because the least input increment is used as the units for manual handle feed and incremental feed, the same value represents a different distance depending on whether the metric or inch input system is used.
> 2 For an axis under diameter programming, the tool moves by the diameter value.

Handle-synchronous feed signal HREV <Gn023.4>

[Classification] Input signal

[Function] This signal selects handle-synchronous feed. That is to say, it causes the cutting feedrate used during automatic operation to be synchronized with the rotation of the manual handle (manual pulse generator).

[Operation] When this signal is set to “1”, the machine tool behaves as follows:
The cutting feedrate used during automatic operation is caused to synchronize with the rotation of the manual pulse generator; instead of being a program-specified feedrate.

- The manual linear/circular interpolation signals MHLCn <Gn544.0-4> and the usage selection of manual linear/circular interpolation signals MHUSn <Gn545.0-4> select the manual pulse generator with which the tool is to be synchronized.
- The feedrate varies with the rotation speed of the manual pulse generator. The travel distance of the tool per pulse of the manual pulse generator (travel distance amount of tangential movement) is determined by the manual handle feed amount selection signals MP1 <Gn019.4> and MP2 <Gn019.5>.

Manual linear/circular interpolation signals
MHLC1 to MHLC5 <Gn544.0 to Gn544.4>

[Classification] Input signal

[Function] Each of these signals selects the manual pulse generator for which the manual linear/circular interpolation function is to be executed. The number at the end of the signal name corresponds to the manual pulse generator number.

[Operation] When this signal is set to “0”

- Manual linear/circular interpolation is disabled. The manual pulse generator is used as a regular manual handle.

When this signal is set to “1”

- Manual linear/circular interpolation is enabled. The usage of the manual handle is selected by the usage selection of manual linear /circular interpolation signals MHUSn <Gn545.0-4>.

Usage selection of manual linear/circular interpolation signal
MHUS1 to MHUS5 <Gn545.0 to Gn545.4>

[Classification] Input signal

[Function] When a manual linear/circular interpolation signal MHLCn <Gn544.0-4> is set to 1 for the manual linear/circular interpolation function, the usage (guidance handle or approach handle) of the nth manual pulse generator is selected by the MHUSn <Gn545.0-4> signals. The number at the end of the signal name corresponds to the manual pulse generator number.

[Operation] When this signal is set to “0”

- The manual handle is used as an approach handle.

When this signal is set to “1”

- The manual handle is used as a guidance handle.
  If handle-synchronous feed signal HREV <Gn023.4> is set to “1”, this unit can be used for handle-synchronous feed.

Selection of usage of the nth manual handle

MHLCn	MHUSn	Usage
0	-	Not used for handle-synchronous feed (feed axis)
1	0	Not used for handle-synchronous feed (approach handle)
1	1	Used for handle-synchronous feed (guidance handle)

Selecting direction of manual handle rotation signal HDSR <Gn193.3>

[Classification] Input signal

[Function] Selects the effective direction of rotation of the manual pulse generator.

[Operation] When this signal is set to “0”

- Only CW (clockwise) is the effective rotation direction of the manual pulse generator.

When this signal is set to “1”

- Only CCW (counterclockwise) is the effective rotation direction of the manual pulse generator.

> **NOTE**
> This signal is available when bit 5 (HSR) of parameter No. 7106 is set to 1.

Feed zero signal FEED0 <Fn066.2>

[Classification] Output signal

[Function] Indicates that the feedrate command (F command) specifies 0.

[Output cond.] Outputting the feed zero signal FEED0 requires that bit 7 (FC0) of parameter No. 1404 be set to 1. If FC0 is set to “0”, FEED0 remains to be 0, regardless of the conditions described below.

This signal is set to “1” in the following conditions:

- The feedrate command (F command) is reset to 0 when the power switched on or a reset occurs.
- A block containing a feedrate command (F command) specifying 0 is executed during automatic operation.

This signal is set to “0” in the following conditions:

- A block containing a feedrate command (F command) specifying a value other than 0 is executed during automatic operation.
- For the machining center series, the cutting feedrate (except 0) specified in parameter No. 1411 is made valid when the power is switched on or a reset occurs.
- Bit 7 (FC0) of parameter No. 1404 is set to 0.

NOTE

The feed zero signal FEED0 is available when bit 7 (FC0) of parameter No. 1404 is set to 1.

⚠ **CAUTION**

On receiving the feed zero signal FEED0 specifying “1”, the PMC returns a dry run signal or handle-synchronous feed signal. Movement of cutting feed block will remain stopped until these signals become "1".

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn019			MP2	MP1				
Gn023				HREV				
Gn046	DRN							
Gn193					HDSR			
Gn544				MHLC5	MHLC4	MHLC3	MHLC2	MHLC1
Gn545				MHUS5	MHUS4	MHUS3	MHUS2	MHUS1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn066						FEED0		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1404	FC0							

[Input type] Parameter input
[Data type] Bit path

#7 FC0 In automatic operation, when cutting feed block (G01, G02, G03, etc.) with feedrate command (F command) of 0 is commanded:
0: An alarm PS0011 “FEED ZERO (COMMAND)” occurs.
1: An alarm PS0011 does not occur, and the block is executed at a feedrate of 0.

NOTE
1 In inverse time feed (G93) mode, the alarm PS1202, "NO F COMMAND AT G93" is issued irrespective of the setting of this parameter.
2 When bit 6 (CLR) of parameter No.3402 is 1, to change this parameter FC0 from 1 to 0, reset the CNC. Or if CLR is 0, turn off and on the CNC.

	#7	#6	#5	#4	#3	#2	#1	#0
1405					HFR			

[Input type] Parameter input
[Data type] Bit path

#3 HFR Feedrate command in rapid traverse is:
0: Parameter setting value.
1: Synchronized with handle pulse by the handle-synchronous feed function.

NOTE
This parameter is available when the handle-synchronous feed function is available.

	#7	#6	#5	#4	#3	#2	#1	#0
3402		CLR						

[Input type] Parameter input
[Data type] Bit path

#6 CLR Reset key on the MDI panel, external reset signal, reset and rewind signal, and emergency stop signal
0: Cause reset state.
1: Cause clear state.
For the reset and clear states, refer to Appendix in the OPERATOR’S MANUAL.

	#7	#6	#5	#4	#3	#2	#1	#0
7106			HSR					

[Input type] Parameter input
[Data type] Bit path

#5 HSR The direction of manual pulse generator rotation in the handle-synchronous feed function is:
0: effective in both.
1: effective in one direction. The effective direction is selected by selecting direction of manual handle rotation signal HDSR <Gn193.3>.

7113	Manual handle feed magnification m

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 2000
This parameter sets the magnification m when manual handle feed movement selection signals MP1 and MP2 are set to “0” and “1”.

7114	Manual handle feed magnification n

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 2000
This parameter sets the magnification when manual handle feed movement selection signals MP1 and MP2 are set to “1”.

3.6 RIGID TAPPING BY MANUAL HANDLE

Overview

To execute rigid tapping, set rigid mode, then switch to handle mode and move the tapping axis with a manual handle.
Rigid tapping by manual handle is enabled by setting bit 0 (HRG) of parameter No. 5203 to 1.
See the section of the rigid tapping to operate the rigid tapping by manual handle.

Explanation

- Basic procedure

1 Stop the spindle and servo axes, then set MDI mode.
2 Enter and execute the following program:
Example 1)
M series/T series (G code system B, C)
M29 S100 ;
G91 G84 Z0 F1000 ;

T

Example 2)
T series (G code system A)
M29 S100 ;
G84 W0 F1000 ;

The program above is required to determine a screw lead and set rigid tapping mode. In this program, a tapping axis must always be specified. Specify a value that does not operate the tapping axis. G84 (G74) is specified for M series while G84 (G88) is specified for T series.

> ⚠ **WARNING**
> In this MDI programming, never specify commands to position the tool at a drilling position and at point R. Otherwise, the tool moves along an axis.

3 Close the position control loop of the spindle and set rigid tapping mode by executing the entered program. At this time, perform gear switching or other sequences with PMC as with memory operation.

4 After rigid mode is set upon completion of MDI program execution, switch to the handle mode.

> ⚠ **CAUTION**
> At this time, never press the reset key. Otherwise, rigid mode is canceled.

5 To perform rigid tapping, select a tapping axis, and move the tapping axis with the manual handle.

- Cancellation of rigid mode

To cancel rigid mode, specify G80 as the normal rigid tapping. When the reset key is pressed, rigid mode is canceled, but the canned cycle is not canceled.
When the rigid tapping signal RGTAP is to be set to “0” for rigid mode cancellation (when bit 2 (CRG) of parameter No. 5200 is set to 0), the G80 command ends after the rigid tapping signal RGTAP is set to “0”.

- Spindle rotation direction

The rotation direction of the spindle is determined by a specified tapping cycle G code and the setting of bit 1 (HRM) of parameter No. 5203. For example, when the parameter HRM is set to 0 in G84 mode, the spindle makes forward rotations as the tapping axis moves in the minus direction. (When the tapping axis moves in the plus direction, the spindle makes reverse rotations.)

- Arbitrary tapping axis

M

By setting bit 0 (FXY) of parameter No. 5101 to 1, an arbitrary tapping axis can be selected. In this case, specify a G code for plane selection and tapping axis address when rigid mode is set in MDI mode.

T

A tapping axis can be freely selected regardless of the setting of FXY (bit 0 of parameter No. 5101). For the program format for FANUC Series 15, however, it depends on the setting of FXY.

- Specification of G84/specification of M29 and G84 in the same block

In an MDI program for setting rigid mode, G84 can be used as a rigid tapping G code (bit 0 (G84) of parameter No. 5200 is set to 1), or M29 and G84 can be specified in the same block.
Example 1 : G91 G84 Z0 F1000 S100;
Example 2 : G91 G84 Z0 F1000 M29 S100;

- Program format for FANUC Series 15

Bit 1 (FCV) of parameter No. 0001 can be set to 1 to perform rigid tapping using program format for FANUC Series 15.

M

Example 1: G91 G84.2 (G84.3) Z0 F1000 S100; (M series)

T

Example 2: G84.2 W0 F1000 S100; (T series : G code system A)
Example 3: G91 G84.2 Z0 F1000 S100; (T series : G code system B, C)

- Acceleration or deceleration type

As the acceleration or deceleration type and acceleration or deceleration time constant used during rigid tapping by manual handle, those set in the rigid tapping parameter are enabled. These are also enabled during extraction.

- Specification of manual handle feed faster than the rapid traverse rate

Set parameter No.7117 to 0 so that when manual handle feed, which is faster than the rapid traverse rate, is specified, the handle pulses beyond the rapid traverse rate are ignored.

- Selecting a spindle from multiple spindles

When there are multiple spindles, a spindle can be selected by specifying the S command and P command at the same time.

Example :

When the second spindle is selected

```
M29 S100 P2;
G91 G84 Z0 F1000;
```

Limitation

- Excessive error check

In rigid tapping by manual handle, only an excessive error during movement is checked.

- Tool axis direction handle feed

In rigid tapping by manual handle, tool axis direction handle feed is disabled.

- Extraction override

In rigid tapping by manual handle, the extraction override function is disabled.

- Acceleration/deceleration time constant for extraction

In rigid tapping by manual handle, the use of an acceleration/deceleration time constant for extraction is disabled.

- Number of repeats

In MDI programming, never specify K0 and L0, which are used to specify that the number of repeats is 0 and to disable the execution of a G84 block. If K0 or L0 is specified, rigid mode cannot be set.

- Positioning of the tool to a drilling position

When positioning the tool to a drilling position, select the X-axis or Y-axis with the axis selection signal in handle mode. Never use the method of positioning to a drilling position in MDI mode or MEM mode because the tapping axis may operate.

- 3-dimensional rigid tapping

3-dimensional rigid tapping cannot be used for rigid tapping using a manual handle.

- Feed forward

In rigid tapping by manual handle, feed forward is disabled even if bit 2 (RFF) of parameter No. 5203 is set to 1 (feed forward is enabled in rigid tapping).

- Constant surface speed control

If rigid tapping is commanded during constant surface speed control, alarm PS0200, ”ILLEGAL S CODE COMMAND” is issued. Command rigid tapping after canceling constant surface speed control.

- Spindle control with servo motor
Rigid tapping by manual handle cannot be used against the rigid tapping with servo motor.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5203							HRM	HRG

[Input type] Parameter input
[Data type] Bit path

#0 HRG Rigid tapping by the manual handle is:
0: Disabled.
1: Enabled.

#1 HRM When the tapping axis moves in the negative direction during rigid tapping controlled by the manual handle, the direction in which the spindle rotates is determined as follows:
0: In G84 mode, the spindle rotates in a normal direction. In G74 mode, the spindle rotates in reverse.
1: In G84 mode, the spindle rotates in reverse. In G74 mode, the spindle rotates in a normal direction.

Alarm and message

No.	Message	Description
PS0200	ILLEGAL S CODE COMMAND	In the rigid tap, an S value was out of range or was not specified. The parameters Nos. 5241 to 5243 setting is an S value which can be specified for the rigid tap. Correct the parameters or modify the program.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64693EN)	Rigid tapping by manual handle
CONNECTION MANUAL (FUNCTION) (this manual)	Rigid tapping

3.7 I/O Link β*i* MANUAL HANDLE INTERFACE

Overview

This function controls manual handle feed for FANUC SERVO AMPLIFIER βi series I/O Link Option (I/O Link β*i*) by using a manual pulse generator on the host. Pulses from manual pulse generator are transferred from the host to I/O Link β*i* through I/O Link. Still further, this function can control the magnification of output signal of pulses from manual pulse generator by changing the parameter. This function is available on the peripheral control interface.

Explanation

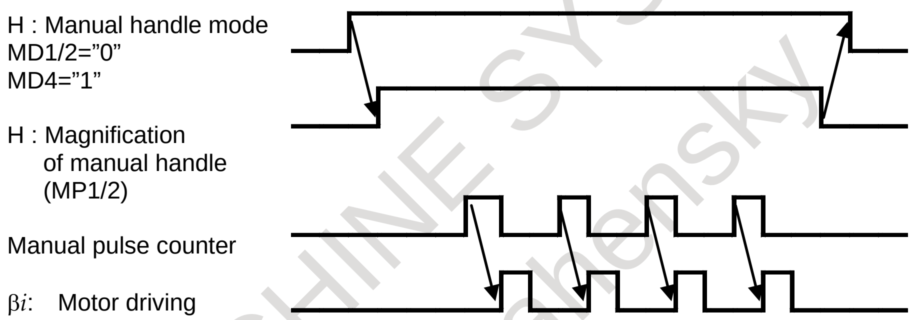
After the manual handle mode of I/O Link β*i* (MD1(Y0.0)=0), MD2(Y0.1)=0, MD4(Y0.2)=1) is selected, the host informs I/O Link β*i* of the magnification of the manual handle generator (MP1(Y7.4), MP2(Y7.5)), and changes the manual pulse counter.

After reading the varied amount of the manual pulse counter, I/O Link β*i* operates the motor.

【Timing chart】

H: The management of the host

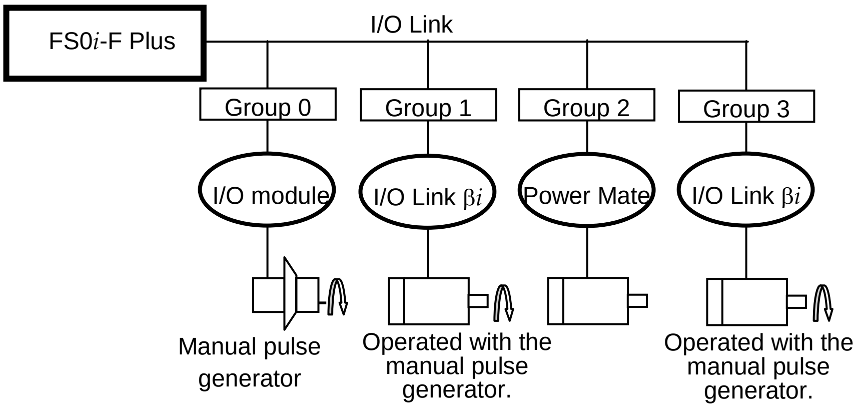
β*i*: The management of the I/O Link β*i*



> ⚠ **CAUTION**
> Be sure to assign 16 bytes signals from the host (CNC) to I/O Link β*i*.

Automatic distinction of an I/O Link β*i*

By setting bit 6 (BHS) of parameter No. 7105 to 1, it is possible to automatically distinguish between an I/O Link β*i* and a Power Mate, so that even if an I/O Link β*i* and a Power Mate are connected at the same time, the I/O Link β*i* can be operated with the manual pulse generator on the host without setting parameters Nos. 12330 to 12337.



Conceptual diagram

NOTE

1 The use of this function requires the Power Mate CNC manager.
2 Do not use direct commands with the I/O Link β*i* that uses this function. (Otherwise, movement along an axis may occur when it is switched between the a direct commands and a peripheral equipment control.)
3 Because of the execution of the automatic distinction to determine whether the slave connected with an I/O Link is an I/O Link β*i* or a Power Mate, the time required until the manual pulse generator can be used becomes longer than usual.
Time delay: (Number of Power Mate connected as slaves × 2) seconds

To execute this automatic distinction, create a ladder program that contains the sequence below.

<1> Check that the β ready signal <F531.7> is on.
<2> Enter manual handle mode.

NOTE

Do not switch to manual handle mode before the β ready signal <F531.7> is turned on.
Otherwise, unintentional movement along an axis may occur.

Signal

Mode selection signal MD1 <Yy+0.0>, MD2 <Yy+0.1>, MD4<Yy+0.2>

[Classification] CNC(host) → I/O Link β*i*

[Function] This signal selects the operation mode of I/O Link β*i*.

[Operation] This signal selects the manual handle mode of the I/O Link β*i*.

MD4	MD2	MD1
1	0	0

⚠ CAUTION

1 Rotate the manual pulse generator on the manual handle mode.
2 Don’t change the operation mode during the manual handle operation of the I/O Link β*i*.

Incremental feed signal MP1 <Yy+7.4>, MP2 <Yy+7.5>

[Classification] CNC(host) → I/O Link β*i*

[Function] This signal selects magnification of manual handle of I/O Link β*i*.

[Operation] I/O Link β*i* drives the motor by pulses multiplied by the magnification of this signal to pulses of the manual pulse generator input during manual handle mode.

MP2	MP1	Distance traveled per a pulse from the manual pulse generator
0	0	User-specified unit × 1
0	1	User-specified unit × 10
1	0	User-specified unit × 100
1	1	User-specified unit × (M/N) (M : Parameter No.0062, N : Parameter No.0063)

> ⚠ **CAUTION**
> 1 These signals are available when the bit 5 (MP) of parameter No. 0005 on the I/O Link β*i* is set to 1.
> 2 These signals are available on the manual handle mode only.
> 3 These signals are shared with the signals for rapid traverse override.
> 4 On manual handle mode, these signals mean Incremental Feed signals. On another mode, these signals mean Rapid Traverse Override signals. When you change the operation mode from the manual handle mode to another, you need to change these signals to the setting of Rapid Traverse Override.

> **NOTE**
> User-specified unit corresponds to increment system in CNC. The machine tool builder defines the weight of one user-specified unit like 0.01mm/0.01deg (IS-A), 0.001mm/0.001deg (IS-B).
> The detection unit is decided from the number of pulses and the amount of the shift per one rotation. The ratio of the user-specified unit to the detection unit is adjusted by Command multiplier (CMR).
> The number of pulses per one rotation and CMR is set to the parameters on the I/O Linkβ*i*.

Manual handle generators selection signal IOLBH1 <G199.0>, IOLBH2 <G199.1>

[Classification] Input signal (common among paths)
[Function] It selects the manual handle generator which transfers to the I/O Link β*i*.
[Operation] It selects the manual handle generator which transfers to the I/O Link β*i* by this signal.

IOLBH2	IOLBH1	Manual handle generator which transfers to I/O Link β*i*
0	0	First unit
0	1	Second unit
1	0	Third unit

> ⚠ **CAUTION**
> Don't change the manual handle generator on the manual handle mode.

β ready signal IOLBR <F531.7>

[Classification] Output signal
[Function] In connection with all I/O Link channels, this signal indicates that the detection of all I/O Link β*i*s are completed.
[Operation] This signal is set to 1 if:

- The detection of all I/O Link β*i*s from those with the same model IDs for all I/O Link channels is completed.

This signal is set to 0 if:

- The detection of all I/O Link β*i*s are not completed.
- The Power Mate CNC Manager is not available.

Signals on I/O Link β*i* side

No.	#7	#6	#5	#4	#3	#2	#1	#0
Yy+0						MD4	MD2	MD1
Yy+7			MP2	MP1				

Signals on CNC side

No.	#7	#6	#5	#4	#3	#2	#1	#0
G199							IOLBH2	IOLBH1
F531	IOLBR							

Parameter

- Parameter on I/O Link β*i* side

No.	#7	#6	#5	#4	#3	#2	#1	#0
0005			MP	IOH				

[Data type] Bit

#4 IOH Manual handle feed by way of I/O Link is :
0 : Unavailable
1 : Available

> **NOTE**
> Be sure to set bit 6 (EXPLS) of parameter No. 0003 on the I/O Link β*i* to 0, when IOH is set to 1.

#5 MP On the manual handle feed, 4 kinds of magnifications set by the incremental feed signal MP1/MP2 are :
0 : Unavailable
1 : Available

- Parameter on CNC side

> ⚠ **CAUTION**
> The Change of the bit 5 (LBH) of parameter No. 7105 and the parameters Nos. 12330 to 12337 is valid after you turn off the power and turn on again. For example, even if you change the bit 5 (LBH) of parameter No. 7105 from 1(available) to 0(unavailable), the motor moves according the movement of the manual handle, till you turn off the power.

	#7	#6	#5	#4	#3	#2	#1	#0
7105		BHS	LBH					

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#5 LBH Manual handle feed for the I/O Link β*i* using the I/O link manual pulse generator is:
0: Disabled.
1: Enabled.

 # 6 BHS When the I/O Link β*i* is operated using the manual pulse generator on the host, automatic determination is:
0: Not used. (Set by parameters Nos. 12330 to 12337.)
1: Used.

Parameter	#7	#6	#5	#4	#3	#2	#1	#0
12330	G17	G16	G15	G14	G13	G12	G11	G10
12331	G1F	G1E	G1D	G1C	G1B	G1A	G19	G18
12332	G27	G26	G25	G24	G23	G22	G21	G20
12333	G2F	G2E	G2D	G2C	G2B	G2A	G29	G28
12334	G37	G36	G35	G34	G33	G32	G31	G30
12335	G3F	G3E	G3D	G3C	G3B	G3A	G39	G38
12336	G47	G46	G45	G44	G43	G42	G41	G40
12337	G4F	G4E	G4D	G4C	G4B	G4A	G49	G48

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

G10 to G4F When the Power Mate or I/O Link β*i* is connected to the I/O Link, these bits set whether to transfer pulses from manual pulse generators connected to the I/O Link to the Power Mate or I/O Link β*i*.

The setting of each bit has the following meaning:
0: Pulses are transferred.
1: Pulses are not transferred.

The bits and the corresponding I/O Link channel numbers and group numbers are listed below:

Parameter	Channel number	Group number
G10	1	0
G11	1	1
G12	1	2
:	:	:
G1F	1	15
:	:	:
G4F	4	15

NOTE
1 When target group is neither Power Mate nor I/O Link β*i*, pulses from the manual pulse generator is not transferred to the group regardless of the parameter setting.
2 When target group is Power Mate, please set the related parameter to 1(not transferred).

Reference item

Manual name	Item name
FANUC SERVO AMPLIFIER β*i* series I/O Link Option MAINTENANCE MANUAL(B-65395EN)	MANUAL HANDLE INTERFACE

3.8 3-DIMENSIONAL MANUAL FEED(M SERIES)

M

Overview

This function enables the use of the following functions.

- 3-dimensional manual feed
  - Tool axis direction handle feed/tool axis direction JOG feed/tool axis direction incremental feed
  - Tool axis right-angle direction handle feed/tool axis right-angle direction JOG feed/tool axis right-angle direction incremental feed
  - Tool tip center rotation handle feed/tool tip center rotation JOG feed/tool tip center rotation incremental feed
  - Table vertical direction handle feed/table vertical direction JOG feed/table vertical direction incremental feed
  - Table horizontal direction handle feed/table horizontal direction JOG feed/table horizontal direction incremental feed

A handle interrupt can be generated for each handle feed. Handle interrupts work according to the corresponding handle feed specifications described hereinafter unless otherwise noted.

- When a second or subsequent manual handle is used

When using a second or subsequent manual handle in 3-dimensional manual feed, set the number of the handle to be used in parameter No. 12323.
At this time, use the manual handle feed axis selection signals/manual handle interrupt axis selection signals for the set handle.
In the description of this manual, the use of the first manual handle is assumed.
When using a second or subsequent manual handle, read HS1A - HS1E <Gn018.0 to 3, Gn411.0> / HS1IA - HS1IE <Gn041.0 to 3, Gn411.4> as the signals for the handle used.
Example 1:
When performing handle feed with the second manual handle
- Set 2 in parameter No. 12323.
- The manual handle feed axis selection signals to be used are HS2A to HS2E <Gn018.4 to 7, Gn411.1>.

Example 2:
When performing handle feed with the fifth manual handle
- Set 5 in parameter No. 12323.
- The manual handle feed axis selection signals to be used are HS5IA to HS5IE <Gn379.4 to 7, Gn412.4>.

- Selecting a coordinate system when calculating the tool direction

If, in 3-dimensional manual feed, a workpiece coordinate system offset is set for a rotation axis, use bit 0 (CAC) of parameter No. 12319 to select whether to use values in the workpiece or machine coordinate system when calculating the tool direction.

- When the bit 0 (CAC) of parameter No. 12319 = 0:
  Values in the machine coordinate system are used.
  For those parameters for configuring the machine, Nos. 19680 to 19714, that depend on the coordinate on a rotation axis, set the values assumed when the machine coordinate on the rotation axis is 0.

- When the bit 0 (CAC) of parameter No. 12319 = 1:
  Values in the workpiece coordinate system are used.
  For those parameters for configuring the machine, Nos. 19680 to 19714, that depend on the coordinate on a rotation axis, set the values assumed when the workpiece coordinate on the rotation axis is 0.

For the functions below, values in the workpiece coordinate system of a rotation axis are used to calculate the tool direction and, therefore, if a workpiece coordinate system offset is set for a rotation axis, and 3-dimensional manual feed is to be used together with any of the functions below, set bit 0 (CAC) of parameter No. 12319 to 1.

- Tilted working plane indexing

(Example)

Related parameters:
No.19680=2 (Tool rotation type)
No.19682=3 (the master rotation axis (C-axis) is about the Z-axis)
No.19687=2 (the slave rotation axis (B-axis) is about the Y-axis)
No.19697=3 (the reference tool axis direction is the Z-axis direction)
No.19698=0 (angle RA when the reference tool axis direction is tilted)
No.19699=0 (angle RB when the reference tool axis direction is tilted)

Workpiece coordinate system offset:
B=10.0

Example 1:
The tool is assumed to face the Z-axis direction when it is in the following states.
Position in the workpiece coordinate system B= -10.0
Position in the machine coordinate system B=0.0
In this case, values in the machine coordinate system must be used to calculate the tool direction, set bit 0 (CAC) of parameter No. 12319 to 0.

Example 2:
The tool is assumed to face the Z-axis direction when it is in the following states.
Position in the workpiece coordinate system B=0.0
Position in the machine coordinate system B=10.0
In this case, values in the workpiece coordinate system must be used to calculate the tool direction, set bit 0 (CAC) of parameter No. 12319 to 1.

- 3-Dimensional manual feed by handle in JOG mode

Even in JOG feed mode, 3-Dimensional manual feed by handle feed can be enabled using bit 0 (JHD) of parameter No.7100.

- The manual handle interruption screen in Tilted working plane indexing mode

In Tilted working plane indexing mode, when 3-dimensional handle interrupt is executed in X,Y, and Z directions of the feature coordinate system (bit 0 (TWD) of parameter No.12320 is set to 1), it can be selected whether to display the travel distance in the manual handle interruption screen in the workpiece coordinate system or the feature coordinate system, by bit 3 (DMK) of parameter No.5402 and by bit 4 (MDT) of parameter No.12319.

In the case to display the travel distance in the feature coordinate system, set bit 3 (DMK) of parameter No.5402 to 0 and bit 4 (MDT) of parameter No.12319 to 1.

3.8.1 Tool Axis Direction Handle Feed/Tool Axis Direction JOG Feed/Tool Axis Direction Incremental Feed

Overview

In the tool axis direction handle feed, tool axis direction JOG feed, and tool axis direction incremental feed, the tool or table is moved in the tool axis direction.

- Tool axis direction

The tool axis direction that is taken when all the rotation axes for controlling the tool are at an angle of 0 degree is specified in parameters Nos.19697, 19698, and 19699. As the rotation axes for controlling the tool rotate, the tool axis direction changes according to the rotation axis angle.

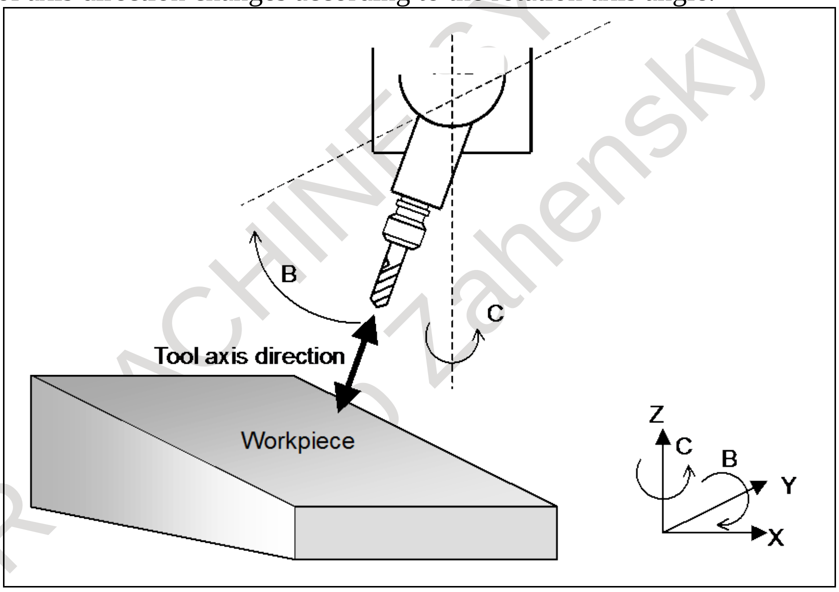


Fig. 3.9.1 Tool axis direction

- Tool axis direction feed in the tilted working plane indexing mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the tool axis direction feed in the tilted working plane indexing mode is assumed to be the Z direction in the feature coordinate system of the tilted working plane indexing.

3.8.1.1 Tool axis direction handle feed

The tool axis direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.

<2> The tool axis direction feed mode signal ALNGH <Gn023.7> is set to 1 and the table base signal TB_BASE <Gn298.0> is set to 0.

<3> The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the tool axis direction handle feed mode effective is set in parameter No. 12310.

<4> The value of parameter No. 12310 matches the first manual handle feed axis selection signals (HS1A - HS1E).

- Amount of movement

When the manual pulse generator is rotated, the tool is moved in the tool axis direction by the amount of rotation.

- Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No. 1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.
If Manual handle feed maximum feedrate change signal HNDLF<Gn023.3> is "1", clamping is performed by the maximum handle feedrate (parameter No. 1434).

3.8.1.2 Tool axis direction JOG feed/tool axis direction incremental feed

The tool axis direction JOG feed or tool axis direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.
<2> The tool axis direction feed mode signal ALNGH is set to 1 and the table base signal TB_BASE is set to 0.
<3> The feed axis direction selection signal (+Jn <Gn100>, -Jn <Gn102> (where n = 1 to the number of controlled axes)) is set to 1 for the axis corresponding to the direction specified by parameter No. 19697. (Even when the tool axis direction is slant because of the settings of parameters Nos.19698 and 19699, the signal that activates the tool axis direction JOG feed or tool axis direction incremental feed is determined by parameter No. 19697 only.)
Ex.) No.19697 = 3 (+Z-axis direction); Z-axis is the 3rd axis of control axes.
- +J3: Tool axis direction +
- -J3: Tool axis direction -

- Feedrate

The feedrate is the dry run rate (parameter No. 1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) for a driven feed axis direction selection signal. The manual feedrate override feature is available.

- Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No. 1424).

3.8.2 Tool Axis Right-Angle Direction Handle Feed/Tool Axis Right-Angle Direction JOG Feed/Tool Axis Right-Angle Direction Incremental Feed

Overview

In the tool axis right-angle direction handle feed, tool axis right-angle direction JOG feed, or tool axis right-angle direction incremental feed, the tool or table is moved in the tool axis right-angle direction.
If bit 1 (FLL) of parameter No. 12320 is set to 1, the tool or table is moved in the latitude or longitude direction determined by the tool axis direction vector.

- Tool axis right-angle direction

There are two tool axis right-angle directions, which are perpendicular to the tool axis direction (see the previous section).

Table 3.8.2 (a) Tool axis direction

Parameter No. 19697	Tool axis right-angle direction 1	Tool axis right-angle direction 2
1 (The reference tool direction is +X.)	+Y direction	+Z direction
2 (The reference tool direction is +Y.)	+Z direction	+X direction
3 (The reference tool direction is +Z.)	+X direction	+Y direction

Fig. 3.9.2 (a) shows the tool axis right-angle directions that may be taken when the angles of all the rotation axes for controlling the tool are 0 degree and when parameters Nos.19698 and 19699 are both set to 0.
When the reference tool axis direction is inclined based on the settings of parameters Nos.19698 and 19699, the tool axis right-angle direction is also inclined as much.
As the rotation axes for controlling the tool rotate, the tool axis right-angle direction changes according to the rotation axis angle.

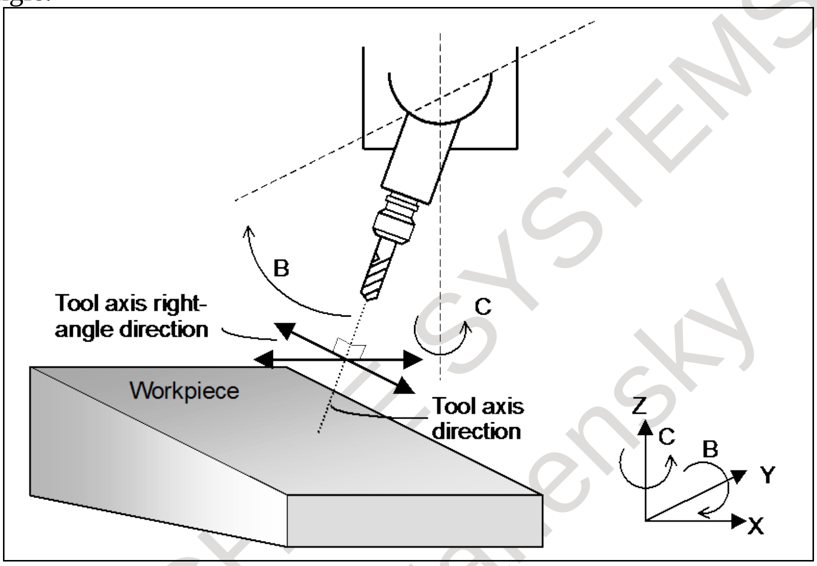


Fig. 3.8.2 (a) Tool axis direction

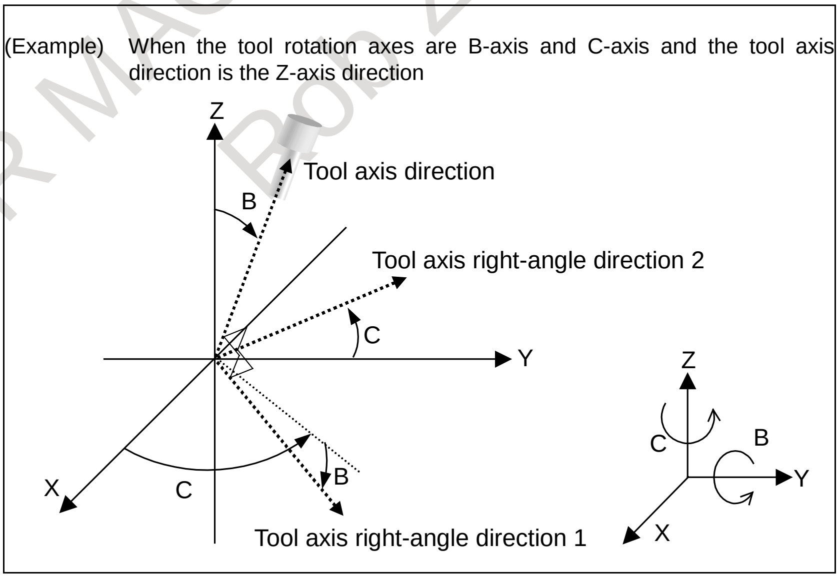


Fig. 3.8.2 (b) Example of tool axis direction

- Latitude and longitude directions

When bit 1 (FLL) of parameter No. 12320 is set to 1, the feed direction is defined as follows:

Let a vector perpendicular to a plane formed by the tool axis direction vector ($\vec{T}$) and normal axis direction vector ($\vec{P}$) (parameter No. 12321) be the tool axis right-angle direction 1 (longitude direction) vector ($\vec{R1}$). When tool axis right-angle direction 1 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Longitude direction feed)

Equation: $\vec{R1} = \vec{P} \times \vec{T}$

Let a vector perpendicular to the tool axis direction vector ($\vec{T}$) and tool axis right-angle direction 1 (longitude direction) vector ($\vec{R1}$) be the tool axis right-angle direction 2 (latitude direction) vector ($\vec{R2}$). When tool axis right-angle direction 2 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Latitude direction)

Equation: $\vec{R2} = \vec{T} \times \vec{R1}$

When the tool axis direction vector ($\vec{T}$) is parallel to the normal axis direction vector ($\vec{P}$) (parameter No. 12321) (when the angle between them is not greater than the setting of parameter No. 12322), tool axis right-angle direction 1 and tool axis right-angle direction 2 are assumed as follows:

Table 3.8.2 (b) Latitude and longitude directions

Parameter No. 12321	Normal axis direction	Tool axis right-angle direction 1	Tool axis right-angle direction 2
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

If 0 is set in parameter No. 12321, the normal axis direction is set to the reference tool axis direction (parameter No. 19697).
If a value other than 0 to 3 is specified in parameter No. 12321, alarm PS5459, "MACHINE PARAMETER INCORRECT" is issued.

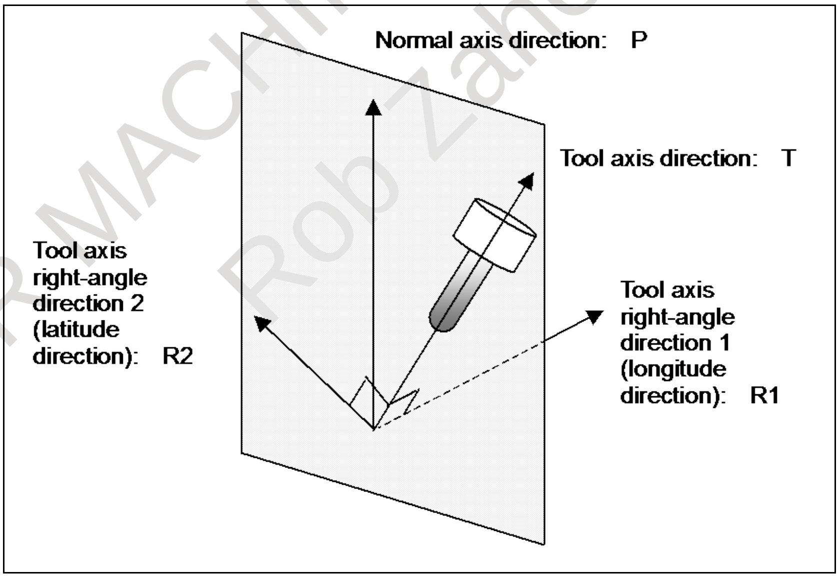


Fig. 3.8.2 (c) Latitude and longitude directions

- Tool axis right-angle direction feed in the tilted working plane indexing mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the tool axis right-angle direction feed in the tilted working plane indexing mode is defined as follows:
Tool axis right-angle direction 1: X direction in the feature coordinate system of the tilted working plane indexing
Tool axis right-angle direction 2: Y direction in the feature coordinate system of the tilted working plane indexing

3.8.2.1 Tool axis right-angle direction handle feed

The tool axis right-angle direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.
<2> The tool axis right-angle direction feed mode signal RGHTH <Gn023.6> is set to 1 and the table base signal TB_BASE is set to 0.
<3> The state of the first manual handle feed axis selection signals (HS1A to HS1E) to make the tool axis right-angle direction handle feed mode effective is set in parameter No. 12311 or 12312.
<4> The value of parameter No.12311 or No.12312 matches the first manual handle feed axis selection signals (HS1A to HS1E).

- Amount of movement

When the manual pulse generator is rotated, the tool is moved in the tool axis right-angle direction by the amount of rotation.

- Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.
If manual handle feed maximum feedrate change signal HNDLF<Gn023.3> is "1", clamping is performed by the maximum handle feedrate (parameter No. 1434).

3.8.2.2 Tool axis right-angle direction JOG feed/tool axis right-angle direction incremental feed

The tool axis right-angle direction JOG feed or tool axis right-angle direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.
<2> The tool axis right-angle direction feed mode signal RGHTH is set to 1 and the table base signal TB_BASE is set to 0.
<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to 1 for the axis corresponding to the direction that is perpendicular to the direction specified by parameter No. 19697. (Even when the tool axis direction is slant because of the settings of parameters Nos.19698 and 19699, the signal that activates the tool axis right-angle direction JOG feed or tool axis right-angle direction incremental feed is determined by parameter No. 19697 only.)
Ex.) Parameter No. 19697=3 (+Z-axis direction); X-, Y-, and Z-axes are the 1st, 2nd, and 3rd axes of control axes respectively.
- +J1: Tool axis right-angle direction 1 +
- -J1: Tool axis right-angle direction 1 -
- +J2: Tool axis right-angle direction 2 +
- -J2: Tool axis right-angle direction 2 -

- Feedrate

The feedrate is the dry run rate (parameter No. 1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) for a driven feed axis direction selection signal. The manual feedrate override feature is available.

- Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No. 1424).

3.8.3 Tool Tip Center Rotation Handle Feed/Tool Tip Center Rotation JOG Feed/Tool Tip Center Rotation Incremental Feed

Overview

In the tool tip center rotation handle feed, tool tip center rotation JOG feed, and tool tip center rotation incremental feed, when a rotary axis is rotated by manual feed, the linear axes (X, Y, and Z axes) are moved so that turning the rotation axis does not change the relative relationship between the tool tip position and the workpiece (table).

- The Fig. 3.8.3 (a) shows an example where the tool is rotated on the rotation axis. In this case, the linear axes are moved so that the position of the tool tip is not moved with respect to the workpiece.

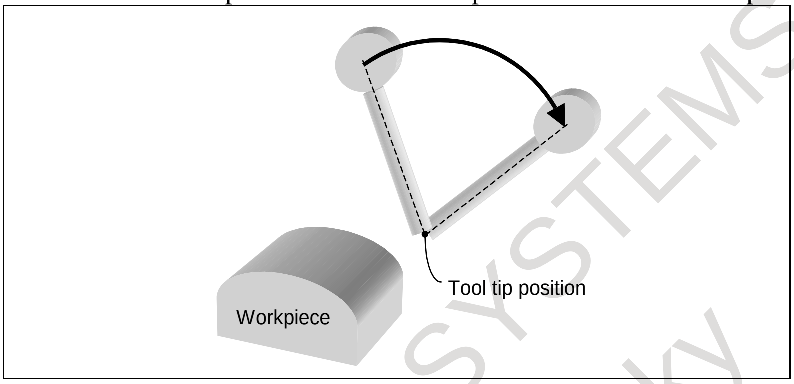


Fig. 3.8.3 (a) Example where the tool is rotated

- The Fig. 3.8.3 (b) shows an example where the table is rotated on the rotation axis. As in the previous case, the linear axes are moved so that the position of the tool tip is not moved with respect to the workpiece (table).

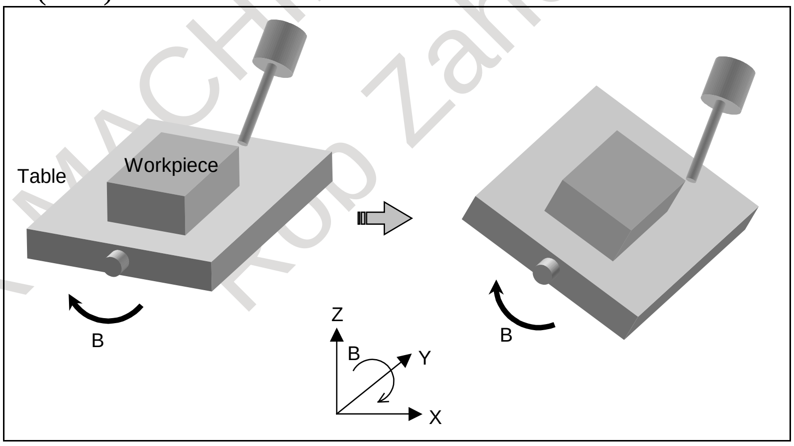


Fig. 3.8.3 (b) Example where the table is rotated

3.8.3.1 Tool tip center rotation handle feed

The tool tip center rotation handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.

<2> The tool tip center rotation feed mode signal RNDH <Gn298.2> is set to 1.

<3> The state of the first manual handle feed axis selection signals (HS1A to HS1E) to make the tool tip center rotation handle feed mode effective is set in parameter No. 12313 or 12314.

<4> The value of parameter No. 12313 or 12314 matches the first manual handle feed axis selection signals (HS1A to HS1E).

- Amount of movement

When the manual pulse generator is rotated, the rotation axis is moved by the amount of rotation. The linear axes (X, Y, and Z axes) are moved so that turning the rotation axis does not change the relative relationship between the tool tip position and the workpiece.

- Feedrate clamp

The feedrate is clamped so that the synthetic speed of the linear axes (the tangential direction speed) does not exceed the manual rapid traverse rate (parameter No. 1424) (of any moving linear axis). The feedrate is also clamped so that the speed of the rotation axis does not exceed the manual rapid traverse rate (parameter No. 1424) (of that particular axis). Handle pulses generated while the clamp feedrate is exceeded are ignored.
If manual handle feed maximum feedrate change signal HNDLF<Gn023.3> is "1", clamping is performed by the maximum handle feedrate (parameter No. 1434).

3.8.3.2 Tool tip center rotation JOG feed/tool tip center rotation incremental feed

The tool tip center rotation JOG feed or tool tip center rotation incremental feed is enabled when the following three conditions are satisfied:
<1> JOG mode or incremental feed mode is selected.
<2> The tool tip center rotation feed mode signal RNDH is set to 1.
<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to 1 for the rotation axis to be rotated.
Ex.) When the B-axis (4th axis) is rotated
- +J4: Tool tip center rotation feed +
- -J4: Tool tip center rotation feed -

- Feedrate

Control is exerted so that the synthetic speed of the linear axes (in the tangential direction) is the dry run rate (parameter No. 1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate of a rotation axis is the jog feedrate (parameter No. 1423) of the axis to be rotated. The manual feedrate override feature is available.

- Feedrate clamp

The feedrate is clamped so that the synthetic speed of the linear axes (the tangential direction speed) does not exceed the manual rapid traverse rate (parameter No. 1424) (of any moving linear axis). The feedrate is also clamped so that the speed of the rotation axis does not exceed the manual rapid traverse rate (parameter No. 1424) (of that particular axis).

3.8.3.3 Selection of the tool length offset value

The tool length in 3-dimensional manual feed is determined as explained below. (Table 3.8.3.3)
If bit 2 (LOD) of parameter No. 19746 is set to 0, the value set in parameter No. 12318 is assumed to be the tool length.
If the LOD parameter is set to 1, and the tool length offset function is performed, the offset data specified for the tool length offset is assumed to be the tool length.
If the LOD parameter is set to 1, and the tool length offset function is not performed, the tool length is determined as follows. If bit 3 (LOZ) of parameter No. 19746 is set to 0, the value set in parameter No. 12318 is assumed to be the tool length in 3-dimensional manual feed; if LOZ is set to 1, the tool length is assumed to be 0.

Table 3.8.3.3 Tool length offset value in 3-dimensional manual feed

<table>
<tr><th colspan="2" rowspan="3"></th><th colspan="3">Bit 2 (LOD) of parameter No.19746</th></tr>
<tr><th rowspan="2">= 0</th><th colspan="2">= 1</th></tr>
<tr><th>Tool length offset enabled</th><th>Tool length offset canceled</th></tr>
<tr><td rowspan="2">Bit 3 (LOZ) of parameter No. 19746</td><td>= 0</td><td rowspan="2">Parameter No. 12318</td><td rowspan="2">Offset data</td><td>Parameter No. 12318</td></tr>
<tr><td>= 1</td><td>0</td></tr>
</table>

The tool length offset function is enabled when the following two conditions are both satisfied:

- The tool length offset function listed below is enabled (modal code of group 8 is except G49)
  - G43/G44 : Tool length compensation
- The H/D code is other than 0.

If bit 6 (CLR) of parameter No. 3402 is set to 0 not to clear the tool length offset vector, G codes of group 8, and H codes at the time of a reset, the tool length offset status is maintained when a reset is made in the tool length offset mode.

3.8.4 Table Vertical Direction Handle Feed/Table Vertical Direction JOG Feed/Table Vertical Direction Incremental Feed

Overview

In the table vertical direction handle feed, table vertical direction JOG feed, and table vertical direction incremental feed, the tool is moved in the table vertical direction.

- Table vertical direction

The table vertical direction is a direction vertical to the table. It is equal to the tool axis direction specified in parameter No. 19697 when all of the rotation axes for controlling the table are at a an angle of 0 degree. When the rotation axes for controlling the table rotate, the table vertical direction changes according to the rotation axis angle.

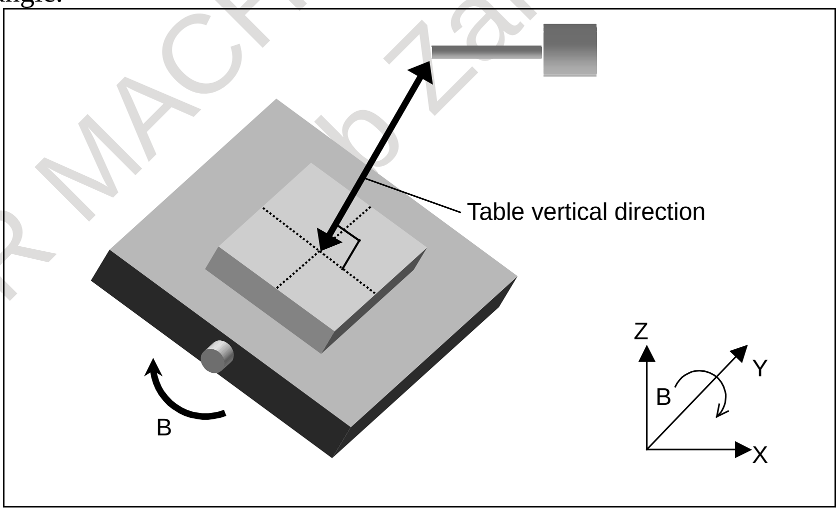


Fig. 3.8.4 Table vertical direction

- Table-based vertical direction feed in the tilted working plane indexing mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the table-based vertical direction feed in the tilted working plane indexing mode is assumed to be the Z direction in the feature coordinate system of the tilted working plane indexing.

3.8.4.1 Table vertical direction handle feed

The table vertical direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.

<2> Both the tool axis direction feed mode signal ALNGH and the table base signal TB_BASE are set to 1.

<3> The state of the first manual handle feed axis selection signals (HS1A to HS1E) to make the table vertical handle feed mode effective is set in parameter No. 12310.

<4> The value of parameter No. 12310 matches the first manual handle feed axis selection signals (HS1A to HS1E).

- Amount of movement

When the manual pulse generator is rotated, the tool is moved in the table vertical direction by the amount of rotation.

- Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No. 1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.
If manual handle feed maximum feedrate change signal HNDLF<Gn023.3> is "1", clamping is performed by the maximum handle feedrate (parameter No. 1434).

3.8.4.2 Table vertical direction JOG feed/table vertical direction incremental feed

The table vertical direction JOG feed or table vertical direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.

<2> Both the tool axis direction feed mode signal ALNGH and the table base signal TB_BASE are set to 1.

<3> The feed axis direction selection signal [+Jn,-Jn (where n = 1 to the number of controlled axes)] is set to 1 for the axis corresponding to the direction specified by parameter No. 19697.

Ex.) Parameter No. 19697 = 3 (+Z-axis direction); Z-axis is the 3rd axis of control axes.
- +J3: Table vertical direction +
- -J3: Table vertical direction -

- Feedrate

The feedrate is the dry run rate (parameter No. 1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) for a driven feed axis direction selection signal. The manual feedrate override feature is available.

- Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No. 1424).

3.8.5 Table Horizontal Direction Handle Feed/Table Horizontal Direction JOG Feed/Table Horizontal Direction Incremental Feed

Overview

In the table horizontal direction handle feed, table horizontal direction JOG feed, and table horizontal direction incremental feed, the tool is moved in the table horizontal direction.

If bit 1 (FLL) of parameter No. 12320 is set to 1, the tool or table is moved in the latitude or longitude direction determined by the table-based vertical direction vector.

- Table horizontal direction

There are two table horizontal directions, which are perpendicular to the table vertical direction (see the previous section).

Table 3.8.5 (a) Table horizontal direction

Parameter No. 19697	Table horizontal direction 1	Table horizontal direction 2
1 (The reference tool direction is +X.)	+Y direction	+Z direction
2 (The reference tool direction is +Y.)	+Z direction	+X direction
3 (The reference tool direction is +Z.)	+X direction	+Y direction

Table 3.8.5 (a) shows the table horizontal directions that may be taken when the angles of all the rotation axes for controlling the table are 0 degree.
As the rotation axes for controlling the table rotate, the table horizontal direction changes according to the rotation axis angle.

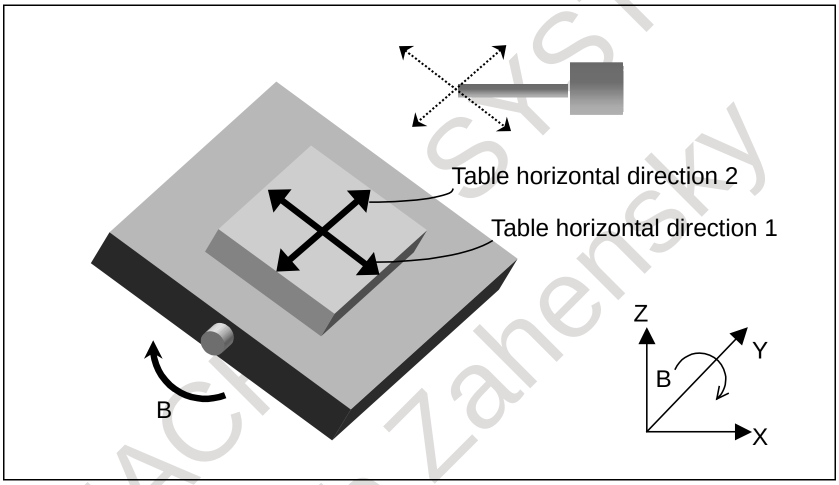


Fig. 3.8.5 (a) Table horizontal direction

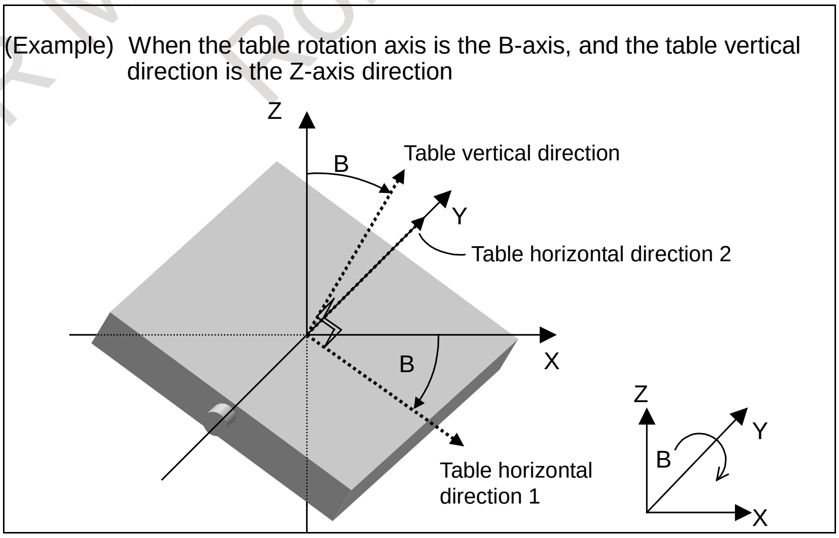


Fig. 3.8.5 (b) Example of table horizontal direction

- Latitude and longitude directions

When bit 1 (FLL) of parameter No. 12320 is set to 1, the feed direction is defined as follows:

Let a vector perpendicular to a plane formed by the table vertical direction vector ($\vec{T}$) and normal axis direction vector ($\vec{P}$) (parameter No. 12321) be the table horizontal direction 1 (longitude direction) vector ($\vec{R}1$). When table horizontal direction 1 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Longitude direction feed)

Equation: $\vec{R}1 = \vec{P} \times \vec{T}$

Let a vector perpendicular to the table vertical direction vector ($\vec{T}$) and table horizontal direction 1 (longitude direction) vector ($\vec{R}1$) be the table horizontal direction 2 (latitude direction) vector ($\vec{R}2$). When table horizontal direction 2 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Latitude direction)

Equation: $\vec{R}2 = \vec{T} \times \vec{R}1$

When table vertical direction vector ($\vec{T}$) is parallel to the normal axis direction vector ($\vec{P}$) (parameter No. 12321) (when the angle between them is not greater than the setting of parameter No. 12322), table horizontal direction 1 and table horizontal direction 2 are assumed as follows:

Table 3.8.5 (b) Latitude and longitude directions

Parameter No. 12321	Normal axis direction	Table horizontal direction 1	Table horizontal direction 2
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

If 0 is set in parameter No. 12321, the normal axis direction is set to the tool axis direction.
If a value other than 0 to 3 is specified in parameter No. 12321, alarm PS5459 is issued.

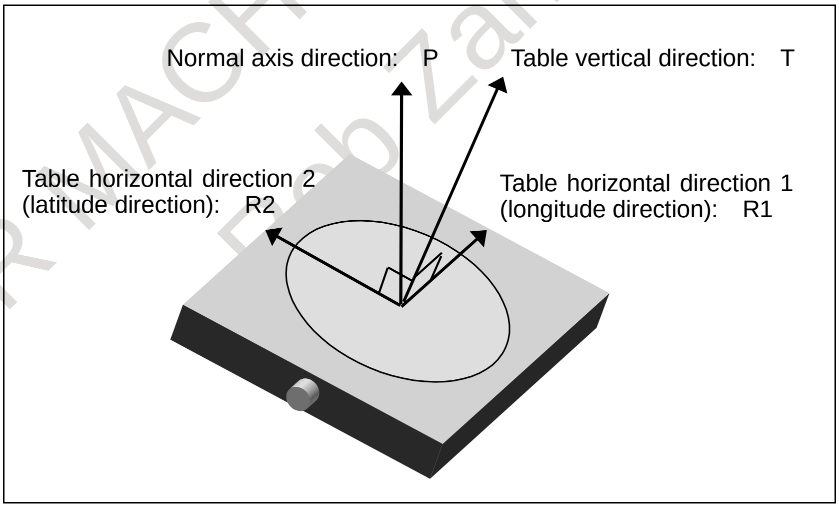


Fig. 3.8.5 (c) Latitude and longitude directions

- Table horizontal direction feed in the tilted working plane indexing mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the table horizontal direction feed in the tilted working plane indexing mode is defined as follows:

Table horizontal direction 1:
X direction in the feature coordinate system of the tilted working plane indexing
Table horizontal direction 2:
Y direction in the feature coordinate system of the tilted working plane indexing

3.8.5.1 Table horizontal direction handle feed

The table horizontal direction handle feed is enabled when the following four conditions are satisfied:
<1> Handle mode is selected.
<2> Both the tool axis right-angle direction feed mode signal RGHTH and the table base signal TB_BASE are set to 1.
<3> The state of the first manual handle feed axis selection signals (HS1A to HS1E) to make the table horizontal direction handle feed mode effective is set in parameter No. 12311 or 12312.
<4> The value of parameter No.12311 or 12312 matches the first manual handle feed axis selection signals (HS1A to HS1E).

- Amount of movement

When the manual pulse generator is rotated, the tool is moved in the table horizontal direction by the amount of rotation.

- Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No. 1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.
If manual handle feed maximum feedrate change signal HNDLF<Gn023.3> is "1", clamping is performed by the maximum handle feedrate (parameter No. 1434).

3.8.5.2 Table horizontal direction JOG feed/table horizontal direction incremental feed

The table horizontal direction JOG feed or table horizontal direction incremental feed is enabled when the following three conditions are satisfied:
<1> JOG mode or incremental feed mode is selected.
<2> Both the tool axis right-angle direction feed mode signal RGHTH and the table base signal TB_BASE are set to 1.
<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to 1 for the axis corresponding to the direction that is perpendicular to the direction specified by parameter No. 19697.
Ex.) Parameter No. 19697 = 3 (+Z-axis direction); X-, Y-, and Z-axes are the 1st, 2nd, and 3rd axes of control axes respectively.
- +J1: Table horizontal direction 1 +
- -J1: Table horizontal direction 1 -
- +J2: Table horizontal direction 2 +
- -J2: Table horizontal direction 2 -

- Feedrate

The feedrate is the dry run rate (parameter No. 1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) of a driven feed axis direction selection signal. The manual feedrate override feature is available.

- Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No. 1424).

Signals

Tool axis direction feed mode signal ALNGH <Gn023.7>

[Classification] Input signal

[Function] Selects the tool axis direction handle feed mode, tool axis direction jog feed mode, tool axis direction incremental feed mode, table vertical direction handle feed mode, table vertical direction jog feed mode, or table vertical direction incremental feed mode.

[Operation]
- If this signal is set to “1” and the table base signal is set to “0”:
  In the handle mode, the tool axis direction handle feed mode is selected.
  In the jog feed mode, the tool axis direction jog feed mode is selected.
  In the incremental feed mode, the tool axis direction incremental feed mode is selected.
- If this signal is set to “1” and the table base signal is set to “1”:
  In the handle mode, the table vertical direction handle feed mode is selected.
  In the jog feed mode, the table vertical direction jog feed mode is selected.
  In the incremental feed mode, the table vertical direction incremental feed mode is selected.

Tool axis right-angle direction feed mode signal RGHTH <Gn023.6>

[Classification] Input signal

[Function] Selects the tool axis right-angle direction handle feed mode, tool axis right-angle direction jog feed mode, tool axis right-angle direction incremental feed mode, table horizontal direction handle feed mode, table horizontal direction jog feed mode, or table horizontal direction incremental feed mode.

[Operation]
- If this signal is set to “1” and the table base signal is set to “0”:
  In the handle mode, the tool axis right-angle direction handle feed mode is selected.
  In the jog feed mode, the tool axis right-angle direction jog feed mode is selected.
  In the incremental feed mode, the tool axis right-angle direction incremental feed mode is selected.
- If this signal is set to “1” and the table base signal is set to “1”:
  In the handle mode, the table horizontal direction handle feed mode is selected.
  In the jog feed mode, the table horizontal direction jog feed mode is selected.
  In the incremental feed mode, the table horizontal direction incremental feed mode is selected.

Tool tip center rotation feed mode signal RNDH <Gn298.2>

[Classification] Input signal

[Function] Selects the tool tip center rotation handle feed mode, tool tip center rotation jog feed mode, or tool tip center rotation incremental feed mode.

[Operation] If this signal is set to “1”:
In the handle mode, the tool tip center rotation handle mode is selected.
In the jog feed mode, the tool tip center rotation jog feed mode is selected.
In the incremental feed mode, the tool tip center rotation incremental feed mode is selected.

> **NOTE**
> The tool axis direction feed mode signal, the tool axis right-angle direction feed mode signal, and the tool tip center rotation feed mode signal can be set to “1” at the same time. In this case, the actually selected feed mode is determined by the manual handle feed axis selection signals for the first manual handle pulse generator (HS1A to HS1E) when the handle feed mode is set, or by the feed axis direction selection signal (+Jn, -Jn, where n is a number from 1 to the number of controlled axes) when the jog feed or incremental feed mode is set. In case of handle feed, the same value must not be set in parameters Nos. 12310 to 12314 that set the status of the manual handle feed axis selection signals for the first manual handle pulse generator to enable each feed operation. If the same value is set in these parameters, no feed mode is selected. Unless the tool axis direction feed mode signal, the tool axis right-angle direction feed mode signal, and the tool tip center rotation feed mode signal are set to “1” at the same time, the same value may be set in parameters Nos. 12310 to 12314.

Table base signal TB_BASE <Gn298.0>

[Classification] Input signal

[Function] Selects the table vertical direction mode and table horizontal direction mode.

[Operation]
- If this signal is set to “0”:
  The tool axis direction handle feed mode, tool axis direction jog feed mode, tool axis direction incremental feed mode, tool axis right-angle direction handle feed mode, tool axis right-angle direction jog feed mode, and tool axis right-angle direction incremental feed mode can be selected.
- If this signal is set to “1”:
  The table vertical direction handle feed mode, table vertical direction jog feed mode, table vertical direction incremental feed mode, table horizontal direction handle feed mode, table horizontal direction jog feed mode, and table horizontal direction incremental feed mode can be selected.

> **NOTE**
> This signal has no influence on the tool tip center rotation handle feed mode, tool tip center rotation jog feed mode, and tool tip center rotation incremental feed mode.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5402					DMK			

[Input type] Parameter input

[Data type] Bit path

#3 DMK The manual handle interruption screen is displayed:

0: Using the program coordinate system (post-conversion coordinate system).

1: Using the workpiece coordinate system (pre-conversion coordinate system).

This parameter is valid only in 3-dimensional coordinate conversion mode and Tilted working plane indexing mode.

> **NOTE**
> In tilted working plane indexing mode, the setting of this parameter is used only when bit 4 (MDT) of parameter No.12319 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
7100								JHD

[Input type] Parameter input
[Data type] Bit path

#0 JHD Manual handle feed in JOG feed mode or incremental feed in the manual handle feed is:
0: Invalid.
1: Valid.

12310	States of the manual handle feed axis selection signals when tool axis direction handle feed/interrupt and table-based vertical direction handle feed/interrupt are performed

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

This parameter sets the states of the manual handle feed axis selection signal (HS1A to HS1E for the first manual handle) or the manual handle interrupt axis selection signal (HS1IA to HS1IE for the first manual handle) to perform tool axis direction handle feed/interrupt and table-based vertical direction handle feed/interrupt.
The handle for which the signal states are set is determined by parameter No. 12323.

<Table of correspondence with the manual handle feed axis selection signals>
If parameter No. 12323 is set to 1, the states of the manual handle feed axis selection signals or manual handle interrupt axis selection signals for the first manual handle in the 3-dimensional manual feed (handle feed) mode and corresponding parameter settings are listed in the table below. When the first manual handle pulse generator is turned after setting the signals corresponding to the value set in the parameter, operation is performed in the specified mode.
If the value set in the parameter is larger than number of controlled axes, the movement is not generated.

HS1E (HS1IE)	HS1D (HS1ID)	HS1C (HS1IC)	HS1B (HS1IB)	HS1A (HS1IA)	Parameter (No. 12310)
0	0	0	0	1	1
0	0	0	1	0	2
0	0	0	1	1	3
0	0	1	0	0	4
0	0	1	0	1	5
0	0	1	1	0	6
0	0	1	1	1	7
0	1	0	0	0	8
0	1	0	0	1	9
0	1	0	1	0	10
0	1	0	1	1	11
0	1	1	0	0	12
0	1	1	0	1	13
0	1	1	1	0	14
0	1	1	1	1	15
1	0	0	0	0	16
1	0	0	0	1	17
1	0	0	1	0	18

HS1E (HS1IE)	HS1D (HS1ID)	HS1C (HS1IC)	HS1B (HS1IB)	HS1A (HS1IA)	Parameter (No. 12310)
1	0	0	1	1	19
1	0	1	0	0	20
1	0	1	0	1	21
1	0	1	1	0	22
1	0	1	1	1	23
1	1	0	0	0	24

If parameter No. 12323 is set to 2 to 5, replace 1 in HS1A to HS1E and HS1IA to HS1IE above with 2 to 5.

12311	States of the manual handle feed axis selection signals when a movement is made in the first axis direction in tool axis normal direction handle feed/interrupt and table-based horizontal direction handle feed/interrupt

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

This parameter sets the states of the manual handle feed axis selection signals (HS1A to HS1E for the first manual handle) or the manual handle interrupt axis selection signal (HS1IA to HS1IE for the first manual handle) when a movement is made in the first axis direction. (For settings, see "Table of correspondence with the manual handle feed axis selection signals" in the description of parameter No. 12310.)
The handle for which the signal states are set is determined by parameter No. 12323.

The table below indicates the relationships of tool axis directions, first axis directions, and second axis directions.

Parameter No. 19697	Tool axis directions	First axis directions	Second axis directions
1	X	Y	Z
2	Y	Z	X
3	Z	X	Y

Note, however, that the table above indicates the directions applicable when the angles of all rotation axes are set to 0.
In tool axis direction/tool axis normal direction feed (not table-based), the directions indicated above assume that 0 is set in parameter No. 19698 and No. 19699. When a rotation axis has made a turn or a nonzero value is set in these parameters in tool axis direction/tool axis normal direction feed, the relevant directions are inclined accordingly.

12312	States of the manual handle feed axis selection signals when a movement is made in the second axis direction in tool axis normal direction handle feed/interrupt and table-based horizontal direction handle feed/interrupt

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

This parameter sets the states of the manual handle feed axis selection signals (HS1A to HS1E for the first manual handle) or the manual handle interrupt axis selection signals (HS1IA to HS1IE for the first manual handle) when a movement is made in the second axis direction. (For settings, see "Table of correspondence with the manual handle feed axis selection signals" in the description of parameter No. 12310.)
The handle for which the signal states are set is determined by parameter No. 12323.

12313	States of the manual handle feed axis selection signals when the first rotation axis is turned in tool tip center rotation handle feed/interrupt

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

This parameter sets the states of the manual handle feed axis selection signals (HS1A to HS1E for the first manual handle) or the manual handle interrupt axis selection signals (HS1IA to HS1IE for the first manual handle) when the first rotation axis is turned in tool tip center rotation handle feed or interrupt. (For settings, see "Table of correspondence with the manual handle feed axis selection signals" in the description of parameter No. 12310.)
The handle for which the signal states are set is determined by parameter No. 12323.

12314	States of the manual handle feed axis selection signals when the second rotation axis is turned in tool tip center rotation handle feed/interrupt

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

This parameter sets the states of the manual handle feed axis selection signals (HS1A to HS1E for the first manual handle) or the manual handle interrupt axis selection signals (HS1IA to HS1IE for the first manual handle) when the second rotation axis is turned in tool tip center rotation handle feed or interrupt. (For settings, see "Table of correspondence with the manual handle feed axis selection signals" in the description of parameter No. 12310.)
The handle for which the signal states are set is determined by parameter No. 12323.

12318	Tool length in 3-dimensional machining manual feed

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets a tool length when tool tip center rotation feed is performed with the 3-dimensional machining manual feed function and when the 3-dimensional machining manual feed screen is displayed.

> **NOTE**
> 1 Specify a radius value to set this parameter.
> 2 Do not change this parameter during 3-dimensional manual feed mode.

	#7	#6	#5	#4	#3	#2	#1	#0
12319	PRA			MDT				CAC

[Input type] Parameter input
[Data type] Bit path

#0 CAC If a workpiece coordinate system offset is set for the rotation axis, the coordinate system of the rotation axis used to calculate the 3-dimensional manual feed is:
0: Machine coordinate system.
For those parameters Nos. 19680 to 19714 used to configure the machine that depend on the coordinates of the rotation axis, set the values assumed when the machine coordinates of the rotation axis are 0.
1: Workpiece coordinate system.
For those parameters Nos. 19680 to 19714 used to configure the machine that depend on the coordinates of the rotation axis, set the values assumed when the workpiece coordinates of the rotation axis are 0.

#4 MDT In tilted working plane indexing mode, when 3-dimensional handle interrupt is executed in X,Y, and Z directions of the feature coordinate system, the travel distance displayed in the manual handle interruption screen is:
0: Displayed in the workpiece coordinate system.
1: Displayed according to the setting of bit 3 (DMK) of parameter No.5402.

This parameter is valid only when setting to execute 3-dimensional machining manual feed to X, Y, and Z directions in the feature coordinate system (bit 0 (TWD) of parameter No. 12320 is set to 1).

#7 PRA The feed direction of a 3-dimensional handle interrupt is:
0: Updated when one of the following operation is executed.
- The mode of NC (ex, MEM or HANDLE) is changed.
- Reset is executed.
- The state of manual handle interrupt axis selection signals (HS1IA to HS1ID <Gn041.0 to 3>, HS1IE<Gn411.4> in case of manual pulse generator No.1) is changed.
- 3-dimensional manual feed turns enabled or disabled.
1: Always updated.

	#7	#6	#5	#4	#3	#2	#1	#0
12320						JFR	FLL	TWD

[Input type] Setting input
[Data type] Bit path

#0 TWD The directions of 3-dimensional machining manual feed (other than tool tip center rotation feed) when the tilted working plane indexing is issued are:
0: Same as those not in the tilted working plane indexing. That is, the directions are:
Tool axis normal direction 1 (table-based horizontal direction 1)
Tool axis normal direction 2 (table-based horizontal direction 2)
Tool axis direction (table-based vertical direction)
1: X, Y, and Z directions in the feature coordinate system.

NOTE
Do not change this parameter during 3-dimensional manual feed mode.

#1 FLL The directions of tool axis normal direction feed or table-based horizontal direction feed in the 3-dimensional machining manual feed mode are:
0: Tool axis normal direction 1 (table-based horizontal direction 1) and tool axis normal direction 2 (table-based horizontal direction 2).
1: Longitude direction and latitude direction.

Bit 1 (FLL) of parameter No. 12320	Bit 0 (TWD) of parameter No. 12320	Directions of 3-dimensional machining manual feed
0	0	Conventional directions
0	1	When the tilted working plane indexing is issued: X, Y, and Z directions in the feature coordinate system When the command is not issued: Conventional directions
1	0	Longitude direction and latitude direction
1	1	When the tilted working plane indexing is issued: X, Y, and Z directions in the feature coordinate system When the command is not issued: Longitude direction and latitude direction

NOTE
Do not change this parameter during 3-dimensional manual feed mode.

#2 JFR The feedrate of 3-dimensional machining manual feed (jog feed or incremental feed) is :
0: The dry run rate (parameter No. 1410).
1: The jog feedrate (parameter No. 1423).

NOTE
Do not change this parameter during 3-dimensional manual feed mode.

12321	Normal axis direction

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

When a tilted working plane indexing (G68.3) is issued to perform 3-dimensional machining manual feed in the latitude direction, longitude direction, and tool axis direction, this parameter sets an axis parallel to the normal direction.
1: Positive (+) X-axis direction
2: Positive (+) Y-axis direction
3: Positive (+) Z-axis direction
0: Reference tool axis direction (parameter No. 19697)

12322	Angle used to determine whether to assume the tool axis direction to be parallel to the normal direction (parameter No. 12321)

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to 90

When a tilted working plane indexing (G68.3) is issued to perform 3-dimensional machining manual feed in the latitude direction, longitude direction, and tool axis direction, if the angle between the tool axis direction and normal direction (parameter No. 12321) is too small, the tool axis direction is assumed to be parallel to the normal direction (parameter No. 12321). This parameter sets the maximum angle at which the tool axis direction is assumed to be parallel to the normal direction.
When this parameter is set to 0 or a value outside the valid range, it is set to 1 degree.

12323	Number of a manual handle used for 3-dimensional machining manual feed

[Input type] Setting input
[Data type] Byte path
[Valid data range] 0 to 5

When 3-dimensional machining manual feed (handle feed) is performed, set the number of the manual handle to be used.
When the second or third manual handle is used for 3-dimensional machining manual feed, the function for manual handle feed with 2/3 handles is required.
When the fourth or fifth manual handle is used for 3-dimensional machining manual feed, the function for manual handle feed with 4/5 handles is required.
If 0 or the number of an unavailable handle is set, the first handle is assumed.

NOTE
Do not change this parameter during 3-dimensional manual feed mode.

	#7	#6	#5	#4	#3	#2	#1	#0
13112	NTD	NTA						

[Input type] Parameter input
[Data type] Bit path

#6 NTA On the 3-dimensional manual feed screen, a table-based pulse amount is:
0: Displayed.
1: Not displayed.

#7 NTD On the 3-dimensional manual feed screen, a tool axis based pulse amount is:
0: Displayed.
1: Not displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
13113					CFD			CLR

[Input type] Parameter input
[Data type] Bit path

#0 CLR Upon reset, the display of a travel distance by 3-dimensional manual feed is:
0: Not cleared.
1: Cleared.

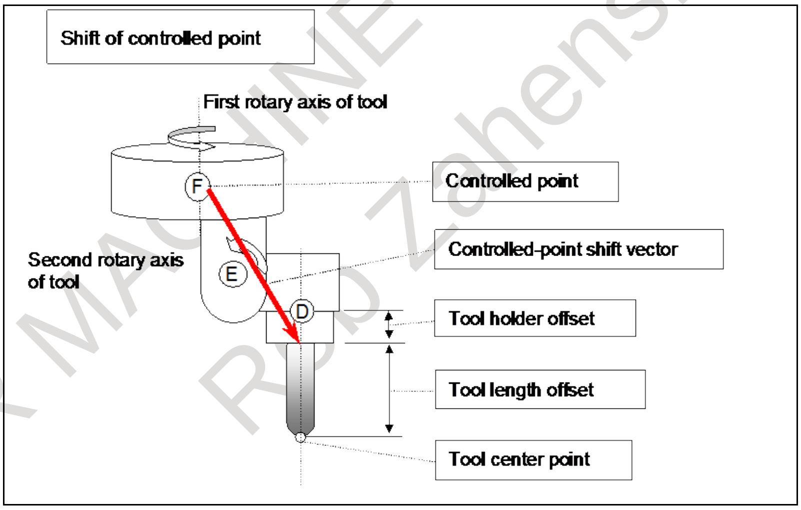
#3 CFD As feedrate F, the 3-dimensional manual feed screen displays:
0: Composite feedrate at the linear axis/rotation axis control point.
1: Feedrate at the tool tip.

	#7	#6	#5	#4	#3	#2	#1	#0
19665			SVC	SPR				

[Input type] Parameter input
[Data type] Bit path

#4 SPR The controlled point is shifted by:
0: Automatic calculation.
1: Using parameter No. 19667.

Bit 5 (SVC) of parameter No. 19665	Bit 4 (SPR) of parameter No. 19665	Shift of controlled point
0	-	Shift is not performed as not done conventionally.
1	0	The controlled point is shifted according to the result of the following automatic calculation: - (Intersection offset vector between the tool axis and the first rotation axis of the tool + intersection offset vector between the second and first rotation axes of the tool + tool holder offset (parameter No. 19666)) (See the figure below.)
1	1	The controlled point is shifted. As the shift vector, the vector set in parameter No. 19667 is used.



[Controlled-point shift vector when automatically calculated]

#5 SVC The controlled point is:
0: Not shifted.
1: Shifted.
The method of shifting is specified by bit 4 (SPR) of parameter No. 19665.

> **NOTE**
> When the machine has no rotation axis for rotating the tool (when parameter No. 19680 is set to 12 to specify the table rotation type), the controlled point is not shifted regardless of the setting of this parameter.

19666	Tool holder offset value

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set an offset value (tool holder offset value) specific to the machine from the control point to the tool attachment position in tool length compensation (after specification of G53.1 in the tilted working plane indexing mode), tool length compensation in tool axis direction, or 3-dimensional manual feed. In tool length compensation (not in the tilted working plane indexing mode), however, tool holder offset can be enabled or disabled with bit 7 (ETH) of parameter No. 19665.

NOTE
Set a radius value.

19667	Controlled-point shift vector

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the shift vector for the controlled point. This value becomes valid when bit 5 (SVC) of parameter No. 19665 is set to 1, and bit 4 (SPR) of parameter No. 19665 is set to 1.

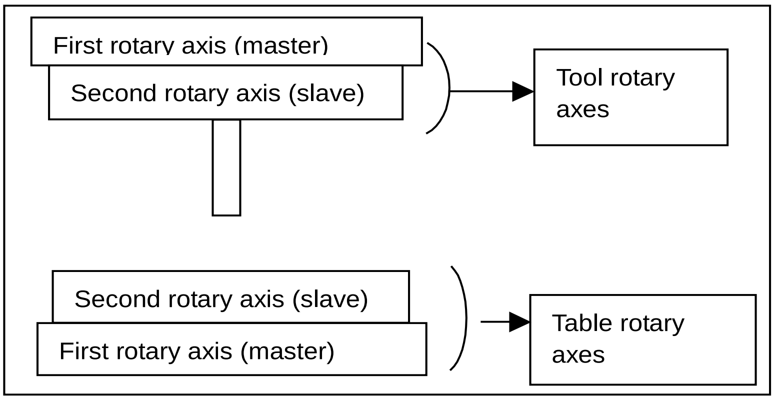
NOTE
Set a radius value.

19680	Mechanical unit type

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 21

Specify the type of the mechanical unit.

Parameter No. 19680	Mechanical unit type	Controlled rotation axis	Master and slave
0		Mechanism having no rotation axis	
2	Tool rotation type	Two rotation axes of the tool	The first rotation axis is the master, and the second rotation axis is the slave.
12	Table rotation type	Two rotation axes of the table	The first rotation axis is the master, and the second rotation axis is the slave.
21	Mixed type	One rotation axis of the tool + one rotation axis of the table	The first rotation axis is the tool rotation axis, and the second rotation axis is the table rotation axis.



NOTE

A hypothetical axis is also counted as a controlled rotary axis.
<Hypothetical axis>
In some cases, it is convenient to use an imaginary rotary axis whose angle is fixed to a certain value. For example, suppose that a tool is mounted in a tilted manner through an attachment. In such a case, the rotary axis considered hypothetically is a hypothetical axis. Bits 0 (IA1) and 1 (IA2) of parameter No. 19696 determine whether each rotary axis is an ordinary rotary axis or a hypothetical axis.

19681	Controlled-axis number for the first rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

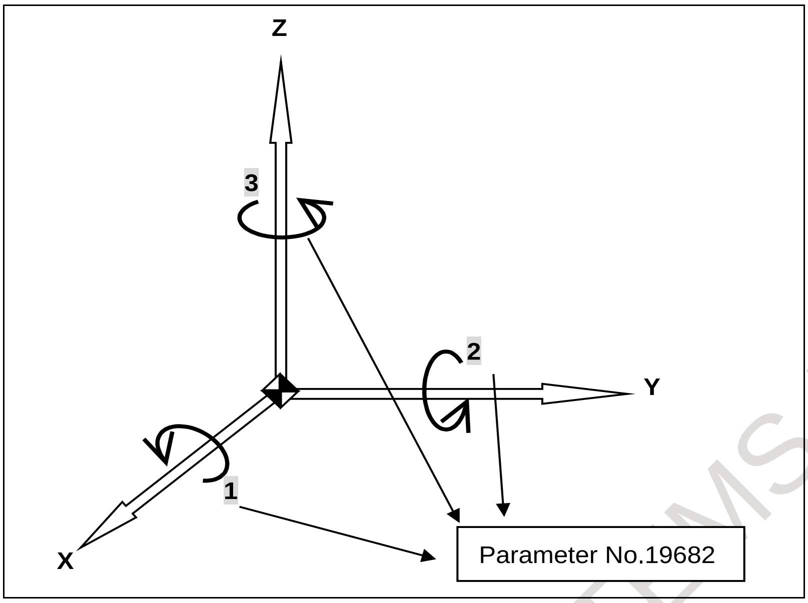
Set the controlled-axis number for the first rotation axis.
For a hypothetical axis (when bit 0 (IA1) of parameter No. 19696 is 1), set 0.

[Example] Assuming that the axis configuration in path 1 is X,Y,Z,B,C and the axis configuration in path 2 is X,Z,C,Y,B, set the parameter to 5 in path 1 and to 3 in path 2 if C is the first rotation axis in each path.

19682	Axis direction of the first rotation axis

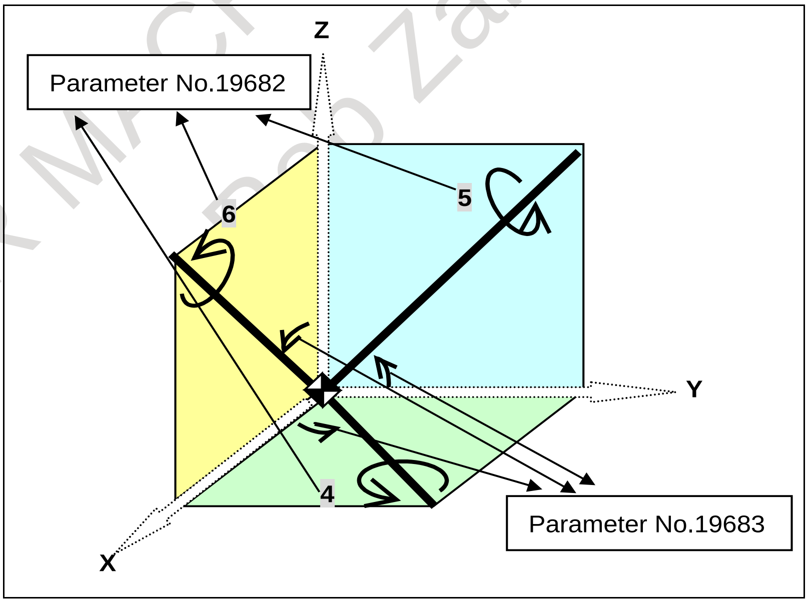
[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

Specify the axis direction of the first rotation axis.
1: On X-axis
2: On Y-axis
3: On Z-axis



19683	Inclination angle when the first rotation axis is an inclined axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] The increment system of the reference axis is to be followed.
[Valid data range] Nine digits of the least input increment (see standard parameter setting table (A).)
(-999999.999 to +999999.999 for IS-B)
When a value 1 to 3 is set in parameter No. 19682, set 0 degrees.
When a value 4 to 6 is set in parameter No. 19682, specify the inclination angle.



19684	Rotation direction of the first rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 1
Set the direction in which the first rotation axis rotates as a mechanical motion when a positive move command is issued.

0: Clockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19682 (right-hand thread rotation)
1: Counterclockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19682 (left-hand thread rotation)

Normally, 0 is set for a tool rotation axis, and 1 is set for a table rotation axis.

19685	Rotation angle when the first rotation axis is a hypothetical axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
When the first rotation axis is a hypothetical axis (bit 0 (IA1) of parameter No. 19696 is 1), set the rotation angle.

19686	Controlled-axis number for the second rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the controlled-axis number for the second rotation axis.
For a hypothetical axis (bit 1 (IA2) of parameter No. 19696 is 1), set 0.
[Example] Assuming that the axis configuration in path 1 is X,Y,Z,B,C and the axis configuration in path 2 is X,Z,C,Y,B, set the parameter to 4 in path 1 and to 5 in path 2 if B is the second rotation axis in each path.

19687	Axis direction of the second rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3
Specify the axis direction of the second rotation axis.
1: On X-axis
2: On Y-axis
3: On Z-axis
When the second rotation axis is the slave axis, the direction when the master axis is at 0 degrees must be set.

19688	Inclination angle when the second rotation axis is inclined

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)
If parameter No. 19687 is set to a value 1 to 3, set 0 degrees.
If parameter No. 19687 is set to a value 4 to 6, set the inclination angle.

19689	Rotation direction of the second rotation axis

[Input type] Parameter input
[Data type] Byte path

[Valid data range] 0 to 1
Set the direction in which the second rotation axis rotates as a mechanical motion when a positive move command is issued.
0: Clockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19687 (right-hand thread rotation)
1: Counterclockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19687 (left-hand thread rotation)
Normally, 0 is set for a tool rotation axis, and 1 is set for a table rotation axis.

19690	Rotation angle when the second rotation axis is a hypothetical axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
When the second rotation axis is a hypothetical axis (bit 1 (IA2) of parameter No. 19696 is 1), set the rotation angle.

	#7	#6	#5	#4	#3	#2	#1	#0
19696							IA2	IA1

[Input type] Parameter input
[Data type] Bit path

#0 IA1 0: The first rotation axis is an ordinary rotation axis.
1: The first rotation axis is a hypothetical axis.
If IA1 is 1, set 0 as the controlled-axis number for the first rotation axis (parameter No. 19681).
Also, set parameters Nos. 19682 to 19685 on the assumption that there is a rotation axis.

#1 IA2 0: The second rotation axis is an ordinary rotation axis.
1: The second rotation axis is a hypothetical axis.
If IA2 is 1, set 0 as the controlled-axis number for the second rotation axis (parameter No. 19686).
Also, set parameters Nos. 19687 to 19690 on the assumption that there is a rotation axis.

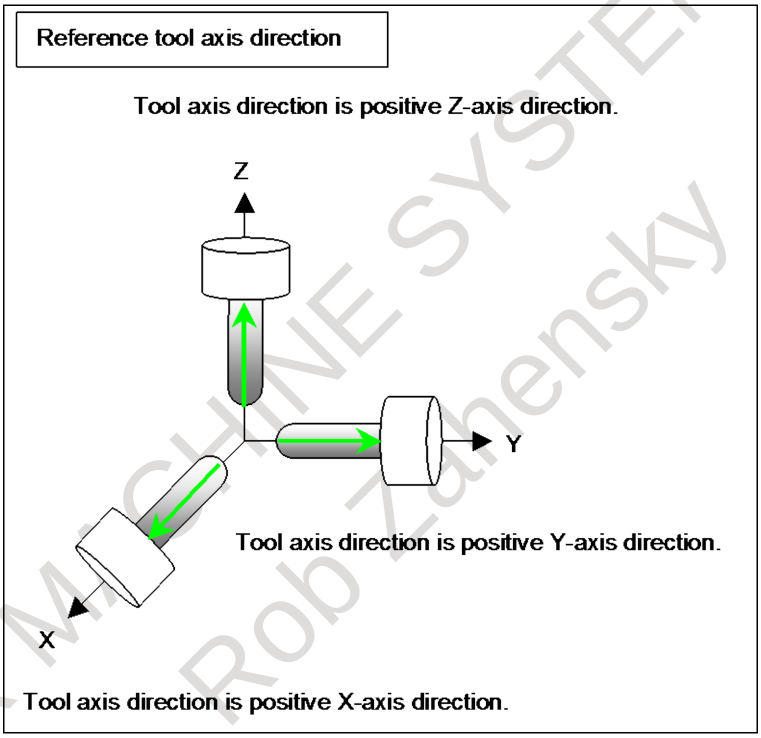
19697	Reference tool axis direction

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

Set the tool axis direction in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees. Also, set the tool axis direction in the machine coordinate system in a mechanism in which only the rotation axes for controlling the table are present (there is no rotation axis for controlling the tool).

1: Positive X-axis direction
2: Positive Y-axis direction
3: Positive Z-axis direction

When the reference tool axis direction is neither the X-, Y-, nor Z-axis direction, set the reference direction in this parameter, then set appropriate angles as the reference angle RA and reference angle RB (parameters Nos. 19698 and 19699).



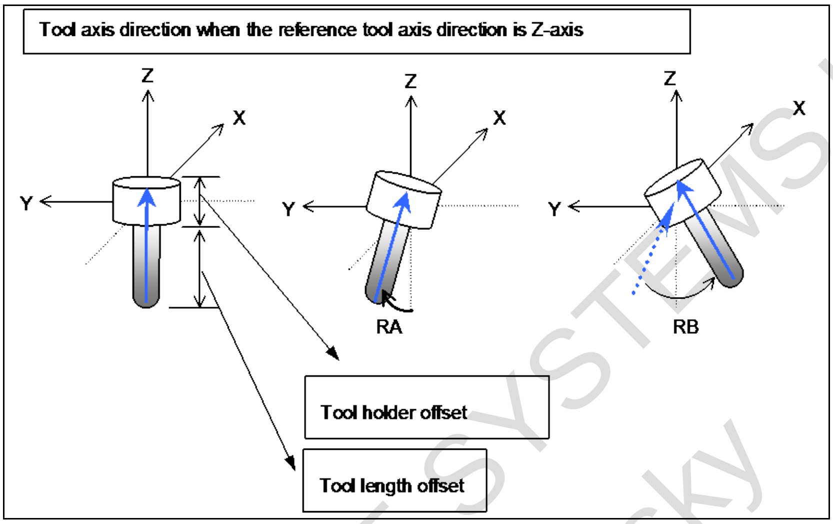
19698	Angle when the reference tool axis direction is tilted (reference angle RA)

19699	Angle when the reference tool axis direction is tilted (reference angle RB)

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)

When the reference tool axis direction (parameter No. 19697) is set to 1, the tool axis is tilted the RA degrees on the Z-axis from the positive X-axis direction to positive Y-axis direction, then the tool axis is tilted the RB degrees on the X-axis from the positive Y-axis direction to positive Z-axis direction.

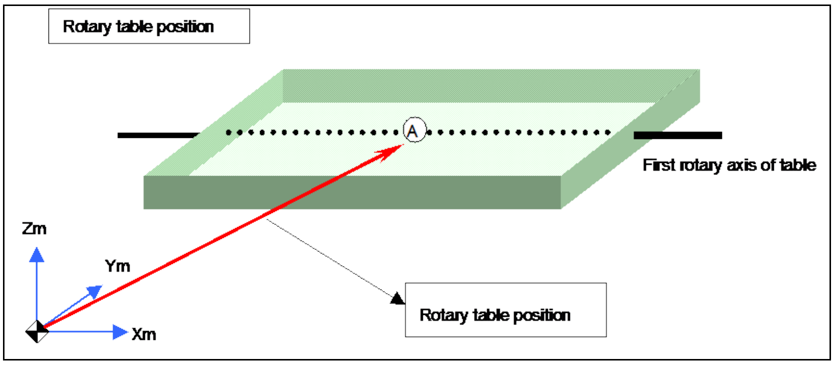
When the reference tool axis direction (parameter No. 19697) is set to 2, the tool axis is tilted the RA degrees on the X-axis from the positive Y-axis direction to positive Z-axis direction, then the tool axis is tilted the RB degrees on the Y-axis from the positive Z-axis direction to positive X-axis direction.
When the reference tool axis direction (parameter No. 19697) is set to 3, the tool axis is tilted the RA degrees on the Y-axis from the positive Z-axis direction to positive X-axis direction, then the tool axis is tilted the RB degrees on the Z-axis from the positive X-axis direction to positive Y-axis direction.



19700	Rotary table position (X-axis of the basic three axes)
19701	Rotary table position (Y-axis of the basic three axes)
19702	Rotary table position (Z-axis of the basic three axes)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set these parameters when parameter No. 19680 is set to 12 or 21. The vector from the origin of the machine coordinate system to point A on the first rotation axis of the table is set as the rotary table position in the machine coordinate system.



NOTE
As point A, set a position that is easy to measure on the first rotary axis of the table.
Set a radius value.
If the rotary table is moved along the X-, Y-, or Z-axis or all of these axes, set the position of the rotary table when the machine coordinates of the X-, Y-, and Z-axes are all set to 0.

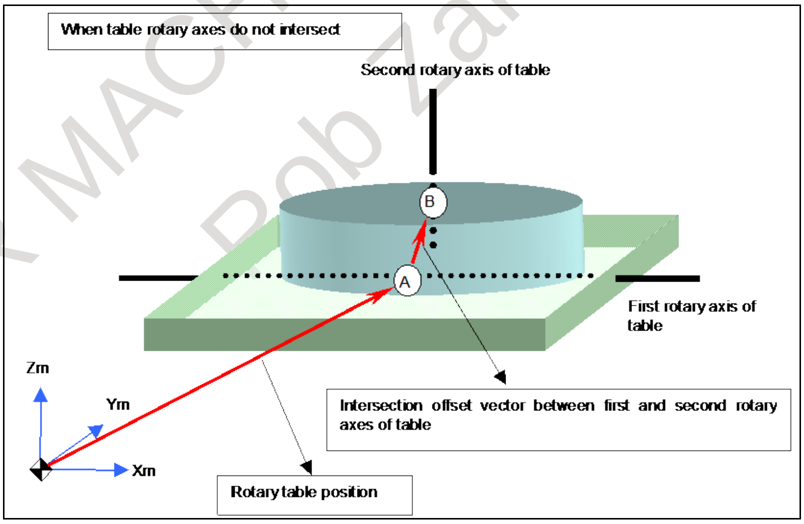
19703	Intersection offset vector between the first and second rotation axes of the table (X-axis of the basic three axes)

19704	Intersection offset vector between the first and second rotation axes of the table (Y-axis of the basic three axes)

19705	Intersection offset vector between the first and second rotation axes of the table (Z-axis of the basic three axes)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set these parameters when the first rotation axis and second rotation axis of the table do not intersect. These parameters are valid when parameter No. 19680 is set to 12. When the rotation axes for controlling the table are all at 0 degrees, the vector from point A to point B on the second rotation axis of the table is set as the intersection offset vector in the machine coordinate system.



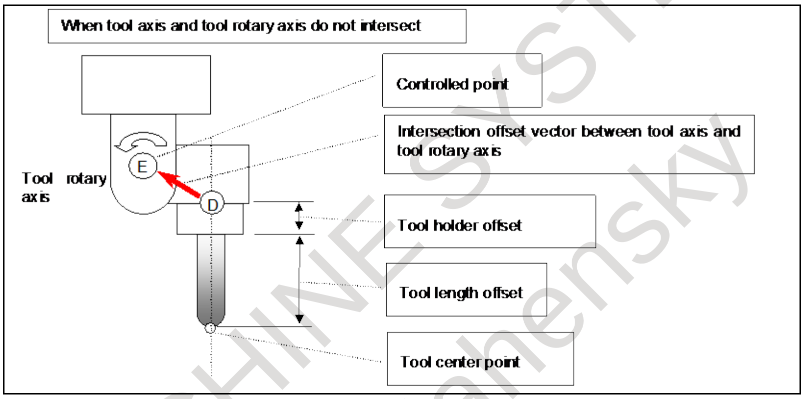
NOTE
As point B, set a position that is easy to measure on the second rotary axis of the table.
Set a radius value.

19709	Intersection offset vector between the tool axis and tool rotation axis (X-axis of the basic three axes)

19710	Intersection offset vector between the tool axis and tool rotation axis (Y-axis of the basic three axes)

19711	Intersection offset vector between the tool axis and tool rotation axis (Z-axis of the basic three axes)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set these parameters when the tool axis and tool rotation axis do not intersect.
These parameters are valid when parameter No. 19680 is set to 2 or 21.
If parameter No. 19680 is 21, set the vector from point D on the tool axis to point E determined on the tool rotation axis as the intersection offset vector in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees.
If parameter No. 19680 is 2, set the vector from point D on the tool axis to point E determined on the second rotation axis of the tool as the intersection offset vector in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees.



NOTE
Point D is determined by adding the tool length offset and tool holder offset (parameter No. 19666) to the tool tip. As point E, set a position that is easy to measure.
Set a radius value.

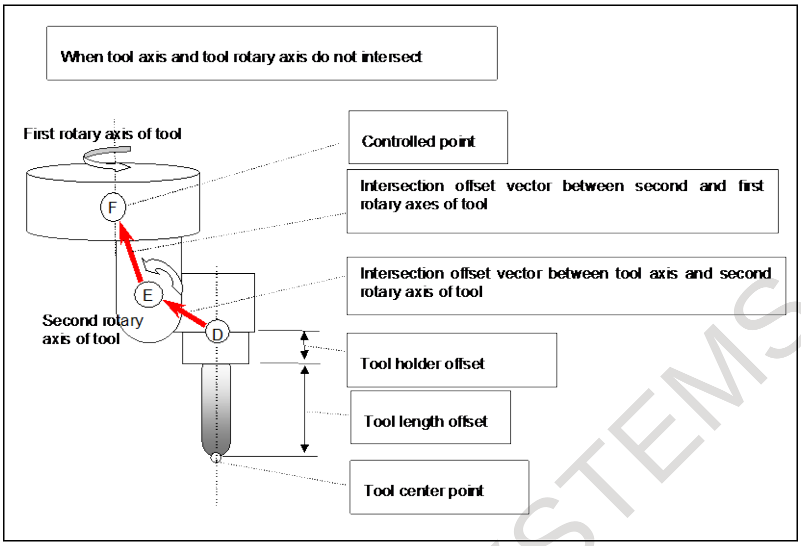
19712	Intersection offset vector between the second and first rotation axes of the tool (X-axis of the basic three axes)

19713	Intersection offset vector between the second and first rotation axes of the tool (Y-axis of the basic three axes)

19714	Intersection offset vector between the second and first rotation axes of the tool (Z-axis of the basic three axes)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set these parameters when the rotation axes of the tool do not intersect.
These parameters are valid when parameter No. 19680 is set to 2.

Set the vector from point E on the second rotation axis of the tool to point F on the first rotation axis of the tool as the intersection offset vector in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees.



> **NOTE**
> As point F, set a position that is easy to measure.
> Set a radius value.

	#7	#6	#5	#4	#3	#2	#1	#0
19746					LOZ	LOD		

[Input type] Parameter input
[Data type] Bit path

#2 LOD As the tool length for 3-dimensional machining manual feed:
0: The value of parameter No. 12318 is used.
1: The tool length currently used for tool length compensation is used.

#3 LOZ When bit 2 (LOD) of parameter No. 19746 is set to 1 and tool length compensation is not applied, as the tool length for 3-dimensional machining manual feed:
0: The value of parameter No. 12318 is used.
1: 0 is used.

Note

> **NOTE**
> 1 To perform a 3-dimensional handle feed, the manual handle feed function is required. To perform a 3-dimensional handle interrupt, the manual handle interrupt function is required.
> 2 When a 3-dimensional handle interrupt is performed, rotation axis command execution must not be in progress in automatic operation.
> 3 When the manual reference position return mode is selected, 3-dimensional manual feed is not enabled.
> 4 If interlock for each axis is enabled to at least one of 3-dimensional manual feed axes, movement with manual feed is not performed.

NOTE

5 When the offset value specified for the tool length offset function is used for tool center point rotation feed (when bit 2 (LOD) of parameter No. 19746 is set to 1), the controlled point should generally be shifted. (Set bit 5 (SVC) of parameter No. 19665 to 1.)
In this case, specify the tool length with a radius value.

6 The feed direction of a 3-dimensional handle interrupt is updated as follows.
(a) When bit 7 (PRA) of parameter No. 12319 is set to 0
It is updated when one of the following operation is executed.
- The mode of NC (ex, MEM or HANDLE) is changed.
- Reset is executed.
- The state of manual handle interrupt axis selection signals (HS1IA to HS1ID <Gn041.0 to 3>, HS1IE<Gn411.4> in case of manual pulse generator No.1) is changed.
- 3-dimensional manual feed turns enabled or disabled.
(b) When bit 7 (PRA) of parameter No. 12319 is set to 1
It is always updated.

Alarm and message

No.	Message	Description
PS5459	MACHINE PARAMETER INCORRECT	A machine configuration parameters Nos.19665 to 19667 or Nos.19680 to 19744 is illegal.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	3-dimensional manual feed

4 REFERENCE POSITION ESTABLISHMENT

4.1 MANUAL REFERENCE POSITION RETURN

Overview

The tool is moved in the direction specified by bit 5 (ZMI) of parameter No. 1006 setting the feed axis and direction select signal to 1 during manual reference position return mode. Movement will continue until the reference position is obtained.
Manual reference position return is performed by using a grid method. The reference position is based on an electrical grid, using on one-rotation signals received from the position detector.

The following signals relate with the manual reference position return:

	Manual reference position return
Mode selection	MD1, MD2, MD4
Selection of reference position return	ZRN, MREF
Selection of axis to be moved	+J1, -J1, +J2, -J2, +J3, -J3, ...
Selection of direction to be moved	
Selection of speed to be moved	ROV1, ROV2
Deceleration signals for reference position return	*DEC1, *DEC2, *DEC3, ...
Completion signals for reference position return	ZP1, ZP2, ZP3, ...
Reference position establishment signals	ZRF1, ZRF2, ZRF3, ...

- Workpiece coordinate system setting/automatic coordinate system setting

When the workpiece coordinate system is enable (bit 0 (NWZ) of parameter No.8136 is 0), the workpiece coordinate system is set upon the completion of manual reference position return.

When the workpiece coordinate system is disable (bit 0 (NWZ) of parameter No.8136 is 0), and bit 0 (ZPR) of parameter No. 1201 is set to 1, the coordinate system is preset automatically after the manual reference position return.
Parameter 1250 can be used to set the workpiece coordinate system, upon the completion of reference position return. The value set in the parameter sets the reference point of the tool holder or the tip position of the reference tool.
This operation is performed as if the following command were specified at the reference position:

G92X$\underline{\alpha}$Y$\underline{\beta}$Z$\underline{\gamma}$;

NOTE
Automatic coordinate system setting is not performed if the workpiece coordinate system is enable (bit 0 (NWZ) of parameter No.8136 is 0). In this case, manual reference position return always establishes a workpiece coordinate system based on the workpiece origin offsets, specified with parameters Nos. 1220 to 1226.

Explanation

- Basic procedure for manual reference position return

<1> Select manual continuous feed (JOG) mode, and the manual reference position return selection signal ZRN to 1.
<2> Feed a target axis toward the reference position by setting an appropriate feed axis and direction selection signal (+J1, -J1, +J2, -J2,…) to 1.

<3> While the feed axis and direction selection signal is 1, rapid traverse takes place along that axis. Although the rapid traverse override signals (ROV1, ROV2) are valid, the override is generally set to 100%.

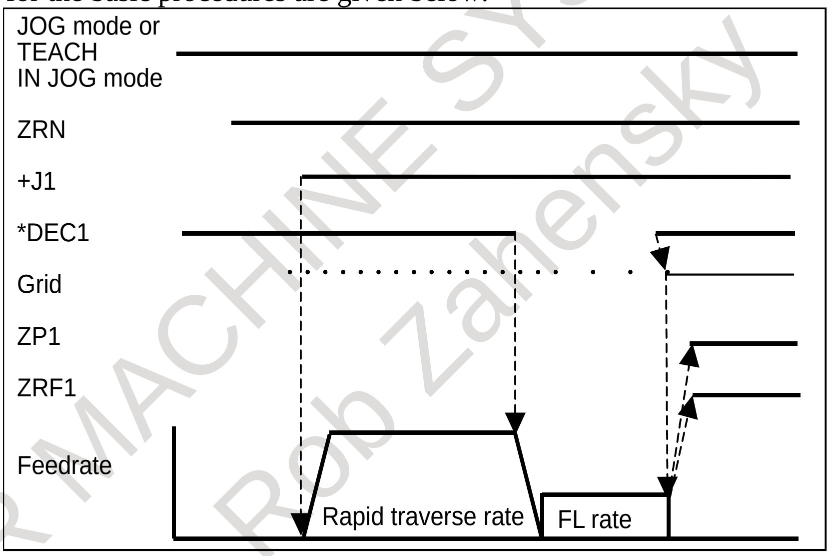
<4> When the reference position is approached, a limit switch installed on the machine is activated, making the deceleration signal for reference position (*DEC1, *DEC2, *DEC3,…) to 0. Consequently, the feedrate is decelerated to 0, then the tool is fed at a constant low speed (reference position return FL feedrate specified by parameter No. 1425 setting).

<5> When the deceleration signal returns to 1 again after the limit switch is passed, the tool is continues to feed, until the tool stops at the first grid point (electric grid point).

<6> Upon confirmation that the current position is at the in-position area, the reference position return end signal (ZP1, ZP2, ZP3,…) and the reference position establishment signal (ZRF1, ZRF2, ZRF3,…) turn to 1.

These steps are repeated for each axis.
The number of simultaneously controlled axes is usually one, but it becomes three by setting bit 0 (JAX) of parameter No. 1002.
If the feed axis direction selection signal (+J1, -J1, +J2, -J2,…) turns to 0 between step (2) and (5) , the tool is stopped at once, and reference position return is canceled. If the signal turn to 1 again, operation resumes from step (3) (rapid traverse).

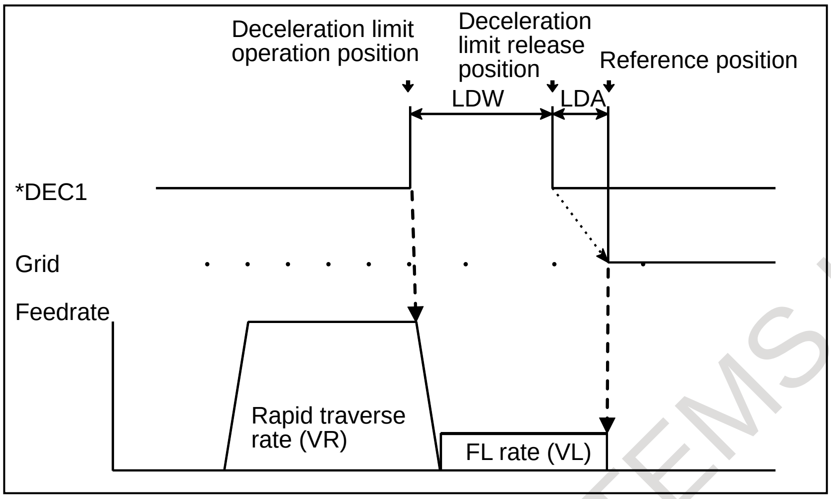
The timing charts for the basic procedures are given below.



NOTE
Grid is not a signal between PMC and CNC.

- Installation conditions for deceleration limit switch

When installing the deceleration limit switch for manual reference position return, ensure that following conditions are satisfied:



(1) LDW : Deceleration dog width [mm, inch]

$$L_{DW} > \frac{V_R\left(\frac{T_R}{2} + 30 + T_S\right) + 4V_L \times T_S}{60 \times 1000}$$

VR : Rapid traverse [mm/min, inch/min]
TR : Rapid traverse time constant [ms]
TS : Servo time constant [ms]
VL : FL speed for reference position return [mm/min, inch/min]

(2) LDA: Distance between deceleration limit switch released position and reference position
LDA = Move amount of 1/2 revolution of motor

Since the above conditions do not include the limit switch operation variations, this point must also be considered at installation.

- Servo position error and one-rotation signal

To perform the first manual reference position return after power on, the tool must be fed in manual reference position return mode, in the reference position return direction at a speed so that the servo position error exceeds the value set in parameter No. 1836. At that time, the tool must cross the grid line corresponding to a one-rotation signal from the position detector.

By the above operation, an electrical grid is established based on one-rotation signals from the position detector.

The servo position error is calculated from the following formula:

$$\text{Servo position error amount} = \frac{F \times 1000}{60} \times \frac{1}{G} \times \frac{1}{U}$$

F : Feedrate [mm/min]
G : Servo loop gain [s-1]
U : Detection unit [μm]

(Example)

When the tool is fed at a feedrate F of 6000 mm/min with a servo loop gain G of 30 [s^{-1}] and a detection unit U of 1 [μm], the servo position error is calculated as follows:

$$\text{Servo position error} = \frac{6000 \times 1000}{60} \times \frac{1}{30} \times \frac{1}{1} = 3333$$

By reversing the formula above, the following formula gives the feedrate F needed to obtain a servo position error of 128, when the servo loop gain G is 30 [s^{-1}] and the detection unit U is 1 [μm]:

$$F = \frac{128 \times 60}{1000} \times 30 = 230 \text{ [mm/min]}$$

Therefore, when the servo loop gain is 30 [s^{-1}], the detection unit is 1 [μm], and parameter No. 1836 is set to 128, the tool must be fed in the reference position return direction at a speed of at least 230 mm/min before completing manual reference position return.

When the one-rotation signal from the position detector is seized at the time of manual reference position return, bit 6 (PCR) of diagnosis data No. 201 is set to 1, and a grid for manual reference position return is established, enabling manual reference position return.

> **NOTE**
> This bit has no effect until the operation of the manual reference position return mode starts.

- Adjusting the reference position

There are the following reference position adjustment methods: Method by grid shift and method by reference position shift.
To shift the reference position within the distance to the first grid point, select the method by grid shift and set bit 4 (SFDx) of parameter No. 1008 to 0.
To shift the reference position beyond the distance to the first grid point, select the method by reference position shift and set bit 4 (SFDx) of parameter No. 1008 to 1.
Either of the methods for adjusting the reference position is available.

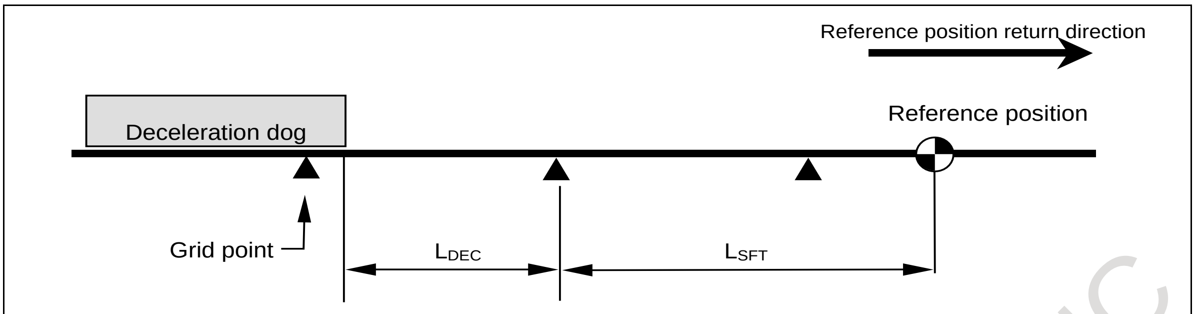
- Adjusting the reference position by grid shift

With the method for shifting the reference position by grid shift, the grid position can be shifted by the distance set in parameter No. 1850.
The distance by which the grid is to be shifted must not exceed the reference counter capacity (parameter No. 1821).
The distance between the tool and the first grid point after the tool leaves the limit switch for deceleration is indicated in diagnosis data No. 302. Furthermore, it is automatically stored in parameter No. 1844.

- Adjusting the reference position by reference position shift

With the method for shifting the reference position by reference position shift, the distance by which the reference position is to be shifted can be set in a parameter to shift it without moving the deceleration dog.

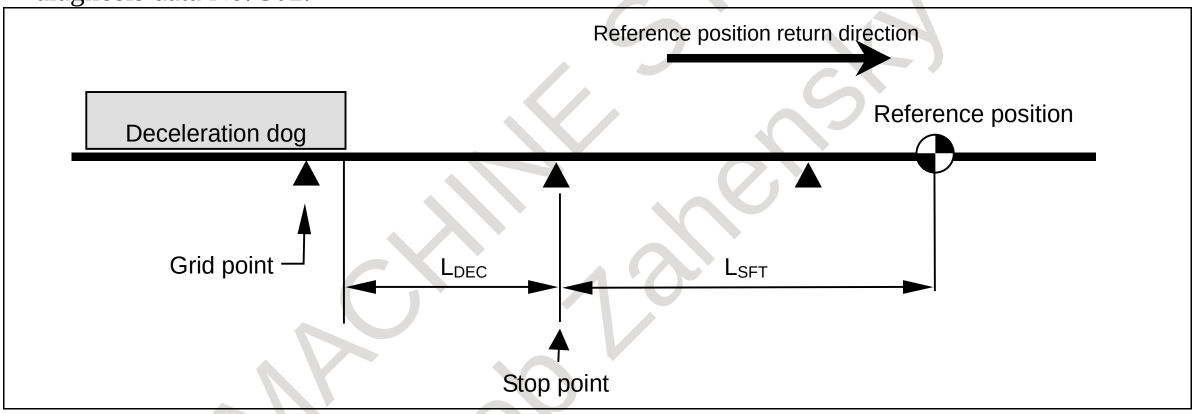
This function is enabled by setting bit 4 (SFDx) of parameter No. 1008.
The distance by which the reference position is to be shifted, shown in the figure below, can be set in parameter No. 1850 to shift the reference position.
The distance L_{DEC} for the axis along which reference position return is performed is automatically stored in parameter No. 1844. The distance L_{DEC} is indicated in diagnosis data No. 302.



L_{DEC}: Distance from the end of the deceleration dog to the first grid point after that when the distance by which the reference position is to be shifted is 0 (parameter No. 1844)
L_{SFT}: Distance by which the reference position is to be shifted (parameter No. 1850)

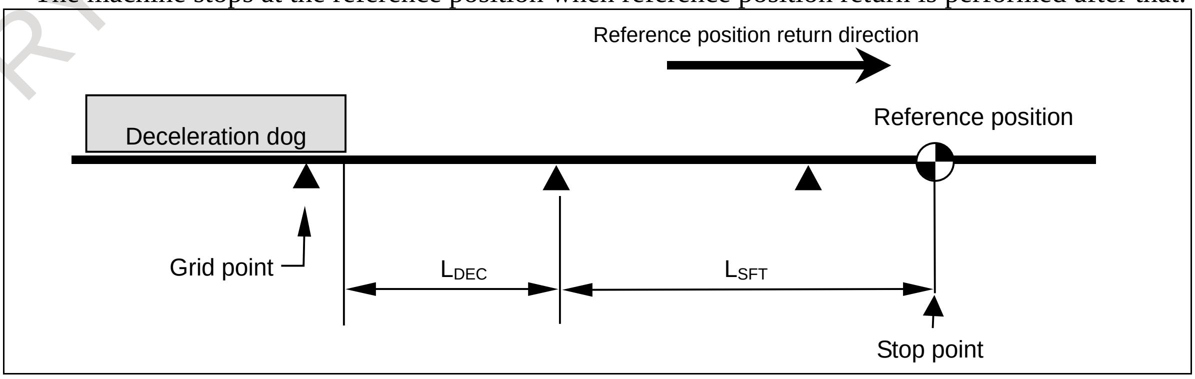
1 Set the following parameter. After setting, turn the power off, then on again.
Set bit 4 (SFDx) of parameter No. 1008 to 1.
The distance from the end of the deceleration dog to the first grid point after that (LDEC (parameter No. 1844)) is 0 when the distance by which the reference position is to be shifted is 0.
Set the distance by which the reference position is to be shifted (parameter No. 1850) to 0.

2 Perform reference position return. The machine stops at the first grid point after leaving the deceleration dog.
The distance LDEC is automatically set in parameter No. 1844. The distance LDEC is indicated in diagnosis data No. 302.



3 Obtain the distance from the stop position to the reference position (distance by which the reference position is to be shifted (LSFT)) and set it in parameter No. 1850.
Then, turn the power off.
Adjusting the reference position is now completed.

4 The machine stops at the reference position when reference position return is performed after that.



> ⚠ **CAUTION**
> 1 Reference position shift is effective only for reference position return using grid points.

> ⚠ **CAUTION**
> 2 Install a deceleration dog so that the distance between the end of the deceleration dog and the first grid point after that is at least 1/2 of the interval between grid points.
> 3 When the distance by which the reference position is to be shifted is 0, the distance between the end of the deceleration dog and the first grid point after that (LDEC (parameter No. 1844)) is automatically set. Do not change the automatically set value.
> 4 This function cannot be used together with reference position setting without dog.
> 5 This function cannot be used together with the grid compensation function in feed axis synchronous control.
> 6 This function cannot be used in case of using method of specifying the reference counter capacity with a fraction (Parameter (No.2179))

- Manual reference position return for a rotation axis about which the tool can rotate one or more turns

For a linear axis, when manual reference position return is performed, the relevant reference position return deceleration signal (*DEC1, *DEC2, *DEC3, ...) is set to 1. At this time, if an electric grid based on one-rotation signals from the position detector is not established, an alarm (PS0090) is issued.

For a rotation axis about which the tool can rotate one or more turns (type A) [bit 0 (ROTx) of parameter No. 1006 is set to 1 and bit 1 (ROSx) of parameter No. 1006 is set to 0], bit 1 (RTLx) of parameter No. 1007 can be set to 1 to specify that the alarm is not issued in this case.
With this setting, manual reference position return may be started with the relevant reference position return deceleration signal (*DEC1, *DEC2, *DEC3, ...) on the machine set to on. In this case, the machine moves not at the reference position return FL feedrate, but at the feedrate set in the relevant parameter.
Since the tool can rotate one or more turns about the rotation axis, it rotates one more turn. When the relevant reference position return deceleration signal (*DEC1, *DEC2, *DEC3, ...) on the machine is turned on, manual reference position return is performed in the same sequence as for normal manual reference position return.

> **NOTE**
> When bit 1 (RTLx) of parameter No. 1007 is set to 0, the relevant reference position return deceleration signal (*DEC1, *DEC2, *DEC3, ...) may be set to 1 before a grid is established. In this case, an alarm (PS0090) is issued.

For a rotation axis about which the tool can rotate up to one turn (type B) [bits 0 (ROTx) and 1 (ROSx) of parameter No. 1006 are set to 1], when the relevant reference position return deceleration signal (*DEC1, *DEC2, *DEC3, ...) is set to 1, a grid may not be established. In this case, an alarm (PS0090) is issued in the same manner as for a linear axis.

- High-speed manual reference position return after the reference position is established

After the reference position is established, the tool can be positioned at the feedrate set in the relevant parameter independently of the deceleration dog for manual reference position return. This operation is enabled by setting bit 7 (SJZ) of parameter No. 0002 and bit 3 (HJZx) of parameter No. 1005.
This operation is called high-speed manual reference position return.

After the reference position is established, setting a feed axis and direction selection signal to 1 in the manual reference position return mode positions the tool at the reference position. This positioning is performed irrespective of the direction specified by the feed axis and direction selection signal. After the completion of the positioning, the relevant reference position return end signal is set to 1.

> ⚠ **CAUTION**
> For high-speed manual reference position return, a feed axis and direction selection signal may be selected in the manual reference position return mode. In this case, the tool may be positioned at the reference position irrespective of the direction specified by the feed axis and direction selection signal, depending on the current position.

If the reference position is lost, manual reference position return using a deceleration dog is performed.

- Coordinate system presetting during high-speed manual reference position return

During high-speed manual reference position return, it is possible to select whether to preset the coordinate system under the conditions below.
The following applies if the workpiece coordinate system is absent and automatic coordinate system setting bit 0 (ZPR) of parameter No. 1201 is 0:
Bit 1 (HZP) of parameter No. 1206 = 0: The coordinate system is preset.
Bit 1 (HZP) of parameter No. 1206 = 1: The coordinate system is not preset.

Signal

Manual reference position return selection signal ZRN<Gn043.7>

[Classification] Input signal

[Function] This signal selects manual reference position return.
Manual reference position return is a kind of jog feed. Therefore, to select manual reference position return, it is required that the jog mode be selected and that the manual reference position return selection signal be set to 1.

[Operation] When the manual reference position return selection signal is set to 1, the control unit performs as described below.

- If jog feed mode is not selected, the control unit ignores the manual reference position return selection signal.
- If jog mode is selected, manual reference position return is enabled. In this case, the manual reference position return selection check signal MREF turns to 1.

> **NOTE**
> If the ZRN status changes state during jog feed, the feedrate is decelerated to 0. Then, to restart reference position return or jog feed, turn feed axis and direction selection signal to 0 then set it to 1.

Manual reference position return selection check signal MREF<Fn004.5>

[Classification] Output signal

[Function] This signal indicates that manual reference position return has been selected.

[Output cond.] This signal turns to 1 when:

- Manual reference position return has been selected.

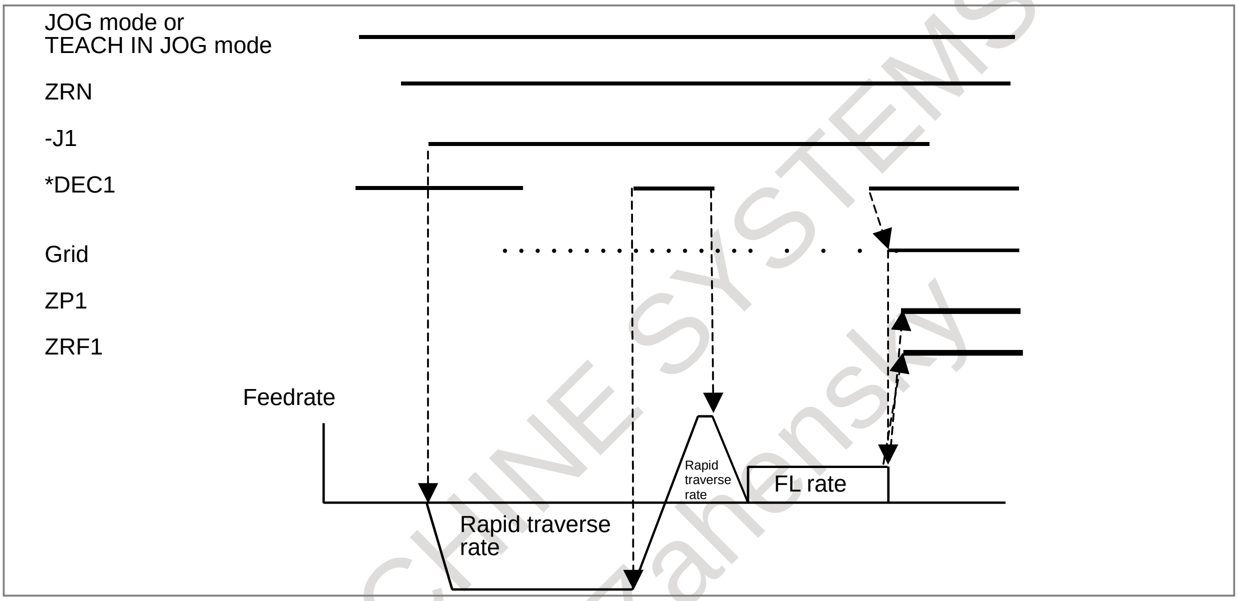
The signal turns to 0 when:

- The selection of manual reference position return has terminated.

Feed axis and direction selection signal

For details about this signal, see "Feed Axis and Direction Selection Signal". Only notes on use of reference position return are given, here.

> **NOTE**
> 1 The direction of reference position return is set for each axis by bit 5 (ZMI) of parameter No. 1006. If the tool is fed opposite to the preset direction in manual reference position return, while the deceleration signal is 0, the tool feeds until the signal returns to 1. The reference position return is then performed automatically in the predetermined direction.



> **NOTE**
> 2 When reference position return is selected, an axis who has already completed referencing movement along that axis is disabled while the reference position return selection signal (ZRN) is 1. To perform movement again, ZRN must be set 0, and the feed axis and direction selection signal must be reset to 0 and then returned to 1 again.

Reference position return deceleration signals *DEC1 to *DEC8<X009>, *DEC1#2 to *DEC8#2<X007>, *DEC1#3 to *DEC8#3<X010>

[Classification] Input signal

[Function] These signals decelerate the feedrate for manual reference position return to a low feedrate (FL).
Deceleration signals are provided for each axis. The number appended to a deceleration signal represents a controlled axis number.
*DECx
x : 1:Reference position return deceleration signal for the first axis
2:Reference position return deceleration signal for the second axis
3:Reference position return deceleration signal for the third axis
:
:

[Operation] For a description of the control unit response to the deceleration signal, see the basic procedure for manual reference position return.

The reference position return deceleration signals are assigned to path 1 to path 3.
For the axes that are assigned to path 1, *DEC1 to *DEC8 (X009) are valid.
For the axes that are assigned to path 2, *DEC1#2 to *DEC8#2 (X007) are valid.
For the axes that are assigned to path 3, *DEC1#3 to *DEC8#3 (X010) are valid.
The reference position return deceleration signals are not assigned to path 4.
(The X address and bit position for these signals for each path can be arbitrarily determined by setting bit 2 (XSG) of parameter No. 3008 to 1.)

Bit 0 (GDC) of parameter No. 3006 can be set to 1 to use input signal Gn196.

> **NOTE**
> Bit 2 (XSG) of parameter No. 3008 can be set to specify whether to fix the address of the reference position return deceleration signals to X009 (path1), X007 (path2), X010 (path3) or use a desired X address.
> When bit 2 (XSG) of parameter No. 3008 is set to 1, the X address and bit position for each reference position return deceleration signal are determined according to the setting of parameters Nos. 3013 and 3014, for each path.

Bit 5 (DEC) of parameter No. 3003 can be set to set the logic for the deceleration signals (*DEC1 to *DEC8).

Reference position return end signals ZP1 to ZP8<Fn094>

[Classification] Output signal

[Function] These signals report that the tool is at the reference position on a controlled axis.
These signals are provided for each axis. The number appended to a signal represents a controlled axis number.

ZPx

x : 1 : Reference position return end signal for the first axis
2 : Reference position return end signal for the second axis
3 : Reference position return end signal for the third axis
:
:

[Output cond.] These signals are set to 1 when:

- Manual reference position returns is completed, and the axis position is in the in-position area.
- Automatic reference position return (G28) is completed, and the axis position is in the in-position area.
- Reference position return check (G27) is completed, and the axis position is in the in-position area.

These signals turn to 0when:

- The tool has moved from the reference position.
- An emergency stop is applied.
- A servo alarm is raised.

Reference position establishment signal ZRF1 to ZRF8<Fn120>

[Classification] Output signal

[Function] Notify the system that the reference position has been established.

A reference position establishment signal is provided for each axis. The number appended to each signal indicates the number of the controlled axis.

ZRFx

x : 1 : 1st-axis reference position establishment signal
2 : 2nd-axis reference position establishment signal
3 : 3rd-axis reference position establishment signal
:
:

[Output cond.] The signals are set to 1 in the following case:

- When the reference position is established after manual reference position return
- When the reference position is established using the absolute-position detector at initial power-on

When the reference position is lost, the signals are set to 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X007	*DEC8[#2]	*DEC7[#2]	*DEC6[#2]	*DEC5[#2]	*DEC4[#2]	*DEC3[#2]	*DEC2[#2]	*DEC1[#2]
X009	*DEC8	*DEC7	*DEC6	*DEC5	*DEC4	*DEC3	*DEC2	*DEC1
X010	*DEC8[#3]	*DEC7[#3]	*DEC6[#3]	*DEC5[#3]	*DEC4[#3]	*DEC3[#3]	*DEC2[#3]	*DEC1[#3]

When the bit 0 (GDC) of parameter No. 3006 is set to 0.

Gn196	*DEC8	*DEC7	*DEC6	*DEC5	*DEC4	*DEC3	*DEC2	*DEC1

When the bit 0 (GDC) of parameter No. 3006 is set to 1.

Gn043	ZRN							

	#7	#6	#5	#4	#3	#2	#1	#0
Fn004			MREF					
Fn094	ZP8	ZP7	ZP6	ZP5	ZP4	ZP3	ZP2	ZP1
Fn120	ZRF8	ZRF7	ZRF6	ZRF5	ZRF4	ZRF3	ZRF2	ZRF1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0002	SJZ							

[Input type] Setting input
[Data type] Bit

#7 SJZ On an axis for which bit 3 (HJZx) of parameter No. 1005 is set:

0: If a reference position is not established yet, reference position return is performed with deceleration dogs.
If a reference position is already established, reference position return is performed at a parameter-set feedrate without using deceleration dogs.

1: Reference position return is performed with deceleration dogs at all times.

NOTE

SJZ is valid for an axis for which bit 3 (HJZx) of parameter No. 1005 is set to 1. When bit 1 (DLZx) of parameter No. 1005 is set to 1, however, manual reference position return after a reference position is set is performed at a parameter-set feedrate, regardless of the setting of SJZ.

	#7	#6	#5	#4	#3	#2	#1	#0
1002					AZR			JAX

[Input type] Parameter input
[Data type] Bit path

#0 JAX Number of axes controlled simultaneously in jog feed, manual rapid traverse and manual reference position return
0: 1 axis
1: 3 axes

#3 AZR When no reference position is set, the G28 command causes:
0: Reference position return using deceleration dogs (as during manual reference position return) to be executed.
1: Alarm PS0304, "G28 IS COMMANDED WITHOUT ZERO RETURN" to be displayed.

NOTE

When reference position return without dogs is specified, (when bit 1 (DLZ) of parameter No. 1005 is set to 1) the G28 command specified before a reference position is set causes an alarm PS0304 to be issued, regardless of the setting of AZR.

	#7	#6	#5	#4	#3	#2	#1	#0
1005					HJZx		DLZx	ZRNx

[Input type] Parameter input
[Data type] Bit axis

#0 ZRNx If a move command other than G28 is specified by automatic operation when no reference position return is performed yet after the power is turned on:
0: The alarm PS0224, "PERFORM REFERENCE POSITION RETURN." is issued.
1: Operation is performed without issuing an alarm.

NOTE

The state in which a reference position has not been established refers to the following state:
- When an absolute position detector is not used and reference position return has not been performed even once after power-up
- When an absolute position detector is used and the association of the machine position with the position detected with the absolute position detector has not been completed (See the description of bit 4 (APZx) of parameter No. 1815.)

#1 DLZx Function for setting the reference position without dogs
- 0: Disabled
- 1: Enabled

#3 HJZx When a reference position is already set:
- 0: Manual reference position return is performed with deceleration dogs.
- 1: Manual reference position return is performed using rapid traverse without deceleration dogs, or manual reference position return is performed with deceleration dogs, depending on the setting of bit 7 (SJZ) of parameter No.0002.

When the function for setting the reference position without dogs (see the description of bit 1 (DLZx) of parameter No. 1005) is used, manual reference position return after a reference position is set is always performed at a parameter-set feedrate, regardless of the setting of HJZ.

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMIx				ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx
#1 ROSx Setting linear or rotary axis.

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No. 1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

#5 ZMIx The direction of manual reference position return is:
0: + direction
1: - direction

	#7	#6	#5	#4	#3	#2	#1	#0
1007				GRDx			ALZx	RTLx

[Input type] Parameter input
[Data type] Bit axis

#0 RTLx When manual reference position return is performed on a rotary axis (A type) with the deceleration dog pressed before a reference position is established:
0: A movement is made at the reference position return feedrate FL.
1: Until a servo motor grid is established, a movement is not made at the reference position return feedrate FL even if the deceleration dog is pressed, but a movement is made at the rapid traverse rate.

If the deceleration dog is released after a movement at the rapid traverse rate and the deceleration dog is then pressed again and released after the rotary axis makes one revolution, reference position return operation is completed.
When this parameter is set to 0, the alarm PS0090, "REFERENCE POSITION RETURN FAILURE" is issued if the deceleration dog is released before a servo motor grid is established.
If this alarm is issued, start manual reference position return at a position sufficiently far away from the reference position.

#1 ALZx In automatic reference position return (G28):
0: Reference position return is performed by positioning (rapid traverse).
If no reference position return is performed after the power is turned on, however, reference position return is performed using the same sequence as for manual reference position return.
1: Reference position return is performed using the same sequence as for manual reference position return.

#4 GRDx When absolute position detection is performed for an axis and the correspondence between the machine position and the position on the absolute-position detector has not yet been established for the axis, reference position setting without digs is:
0: Not performed more than once.
1: Performed more than once.

	#7	#6	#5	#4	#3	#2	#1	#0
1008				SFDx				

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#4 SFDx In reference position return based on the grid method, the reference position shift function is:
0: Disabled
1: Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
1201						ZCL		ZPR

[Input type] Parameter input
[Data type] Bit path

#0 ZPR Automatic setting of a coordinate system when the manual reference position return is performed
0: Not set automatically
1: Set automatically

> **NOTE**
> ZPR is valid while a workpiece coordinate system function is disable (bit 0 (NWZ) of parameter No.8136 is 1). If a workpiece coordinate system function is enable (bit 0 (NWZ) of parameter No.8136 is 0), making a manual reference position return always causes the workpiece coordinate system to be established on the basis of the workpiece zero point offset (parameters Nos. 1220 to 1226), irrespective of this parameter setting.

#2 ZCL Local coordinate system when the manual reference position return is performed
0: The local coordinate system is not canceled.
1: The local coordinate system is canceled.

> **NOTE**
> ZCL is valid when the workpiece coordinate system is enable (bit 0 (NWZ) of parameter No.8136 is 0). In order to use the local coordinate system (G52), enable the workpiece coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
1206							HZP	

[Input type] Parameter input
[Data type] Bit path

#1 HZP At high-speed manual reference position return, presetting the coordinate system is:
0: Performed.
1: Not performed.

> **NOTE**
> This parameter is valid when no workpiece coordinate system exists with bit 0 (ZPR) of parameter No. 1201 set to 0.

1240	Coordinate value of the reference position in the machine coordinate system

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)

[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the reference position in the machine coordinate system.

1250	Coordinate system of the reference position used when automatic coordinate system setting is performed

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate system of the reference position on each axis to be used for performing automatic coordinate system setting.

	#7	#6	#5	#4	#3	#2	#1	#0
1401						JZR		RPD

[Input type] Parameter input
[Data type] Bit path

#0 RPD Manual rapid traverse during the period from power-on time to the completion of the reference position return.
0: Disabled (Jog feed is performed.)
1: Enabled

#2 JZR The manual reference position return at jog feedrate
0: Not performed
1: Performed

	#7	#6	#5	#4	#3	#2	#1	#0
1404							DLF	

[Input type] Parameter input
[Data type] Bit path

#1 DLF After a reference position is set, manual reference position return performed at:
0: Rapid traverse rate (parameter No. 1420)
1: Manual rapid traverse rate (parameter No. 1424)

> **NOTE**
> This parameter selects a feedrate for reference position return performed without dogs. This parameter also selects a feedrate when manual reference position return is performed according to bit 7 (SJZ) of parameter No.0002 using rapid traverse without deceleration dogs after a reference position is set.

1420	Rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)

[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1423	Feedrate in manual continuous feed (jog feed) for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

(1) When bit 4 (JRV) of parameter No. 1402 is set to 0 (feed per minute), specify a jog feedrate (feed per minute) under an override of 100%.
(2) When bit 4 (JRV) of parameter No. 1402 is set to 1 (feed per revolution), specify a jog feedrate (feed per revolution) under an override of 100%.

NOTE
This parameter is clamped to the axis-by-axis manual rapid traverse rate (parameter No. 1424).

1424	Manual rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rate of manual rapid traverse when the rapid traverse override is 100% for each axis.

NOTE
1 If 0 is set, the rate set in parameter No. 1420 (rapid traverse rate for each axis) is assumed.
2 When manual rapid traverse is selected (bit 0 (RPD) of parameter No. 1401 is set to 1), manual feed is performed at the feedrate set in this parameter, regardless of the setting of bit 4 (JRV) of parameter No. 1402.

1425	FL rate of the reference position return for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

1428	Reference position return feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets a rapid traverse rate for reference position return operation using deceleration dogs, or for reference position return operation before a reference position is set.
This parameter is also used to set a feedrate for the rapid traverse command (G00) in automatic operation before a reference position is set.

NOTE

1 To this feedrate setting 100%, a rapid traverse override (F0, 25, 50, or 100%) is applicable.
2 For automatic return after completion of reference position return and machine reference position establishment, the normal rapid traverse rate is used.
3 As a manual rapid traverse rate before machine reference position establishment by reference position return, the jog feedrate or manual rapid traverse rate can be selected with bit 0 (RPD) of parameter No. 1401.

	Before reference position establishment	After reference position establishment
Automatic reference position return (G28)	No. 1428	No. 1420
Automatic rapid traverse (G00)	No. 1428	No. 1420
Manual reference position return *1	No. 1428	No. 1428 *3
Manual rapid traverse	No. 1423 *2	No. 1424

4 When parameter No. 1428 is set to 0, the following parameter-set feedrates are applied.

	Before reference position establishment	After reference position establishment
Automatic reference position return (G28)	No. 1420	No. 1420
Automatic rapid traverse (G00)	No. 1420	No. 1420
Manual reference position return *1	No. 1424	No. 1424 *3
Manual rapid traverse	No. 1423 *2	No. 1424

No. 1420: Rapid traverse rate
No. 1423: Jog feedrate
No. 1424: Manual rapid traverse rate
*1 : By using bit 2 (JZR) of parameter No. 1401, the jog feedrate can be used for manual reference position return at all times.
*2 : When bit 0 (RPD) of parameter No. 1401 is set to 1, the setting of parameter No. 1424 is used.
*3 : When rapid traverse is used for reference position return without dogs or manual reference position return after reference position establishment, regardless of the deceleration dog, the feedrate for manual reference position return based on these functions is used (the setting of bit 1 (DLF) of parameter No. 1404 is followed).

1821	Reference counter size for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

Set a reference counter size.
As a reference counter size, specify a grid interval for reference position return based on the grid method.
When a value less than 0 is set, the specification of 10000 is assumed.

1836	Servo error amount where reference position return is possible

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] 0 to 32767

This parameter sets a servo error used to enable reference position return.
In general, set this parameter to 0. (When 0 is set, 128 is assumed as the default.)
If, during reference position return, such a feedrate as exceeding a set value is not reached even once before the limit switch for deceleration is released (the deceleration signal (*DEC) for reference position return is set to 1 again), the alarm PS0090, "REFERENCE POSITION RETURN FAILURE" is issued.

1844	Distance to the first grid point when the reference position shift amount in the reference position shift function is 0 or when a reference position return is made by grid shift

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999999999 to 999999999

(1) When the reference position shift function is enabled (when bit 4 (SFDx) of parameter No. 1008 is set to 1)
Set the distance (detection unit) to the first grid point from a point at which the deceleration dog is released when the reference position shift (parameter No. 1850) is set to 0.
(2) When a reference position return is made by grid shift with a setting not to use reference position setting without dogs (when bit 4 (SFDx) of parameter No. 1008 is set to 0, and bit 1 (DLZx) of parameter No. 1005 is set to 0)
Set the distance to the first grid point from a point at which the deceleration dog is released. (Detection unit)
(3) When a reference position return is made by grid shift with a setting to use reference position setting without dogs (when bit 4 (SFDx) of parameter No. 1008 is set to 0, and bit 1 (DLZx) of parameter No. 1005 is set to 1)
Set the distance from the start position for reference position setting without dogs to the first grid point. (Detection unit)

NOTE

1. When the reference position shift function is enabled (when bit 4 (SFDx) of parameter No. 1008 is set to 1)
   When bit 4 (SFDx) of parameter No. 1008 is set to 1, the distance from a point at which the deceleration dog is released to the first grid point (parameter No. 1844) is set to 0, and reference position shift (parameter No. 1850) is set to 0, a manual reference position return allows this parameter to be set automatically. Do not change an automatically set value.
2. When a reference position return is made by grid shift with a setting not to use reference position setting without dogs (when bit 4 (SFDx) of parameter No. 1008 is set to 0, and bit 1 (DLZx) of parameter No. 1005 is set to 0)
   When a manual reference position return using deceleration dogs is made, this parameter is set automatically.
3. When a reference position return is made by grid shift with a setting to use reference position setting without dogs (when bit 4 (SFDx) of parameter No. 1008 is set to 0, and bit 1 (DLZx) of parameter No. 1005 is set to 1)
   When a reference position setting without dogs is made, this parameter is set automatically.

1850	Grid shift and reference position shift for each axis

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -99999999 to 99999999

To shift the reference position, the grid can be shifted by the amount set in this parameter. Up to the maximum value counted by the reference counter can be specified as the grid shift.

In case of bit 4 (SFDx) of parameter No. 1008 is 0: Grid shift
In case of bit 4 (SFDx) of parameter No. 1008 is 1: Reference point shift

NOTE

For setting the reference position without dogs, only the grid shift function can be used.
(The reference position shift function cannot be used.)

	#7	#6	#5	#4	#3	#2	#1	#0
3003			DEC					

[Input type] Parameter input
[Data type] Bit path

#5 DEC Deceleration signal (*DEC1 to *DEC8) for reference position return
0: Deceleration is applied when the signal is 0.
1: Deceleration is applied when the signal is 1.

	#7	#6	#5	#4	#3	#2	#1	#0
3006								GDC

[Input type] Parameter input
[Data type] Bit

#0 GDC As the deceleration signal for reference position return *DEC:
0: X0009 is used.
1: Gn196 is used. (X0009 is disabled.)

> ⚠ **WARNING**
> If the signal *DEC<X0009> is used by another usage when you set 0 to this parameter, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used X address.

	#7	#6	#5	#4	#3	#2	#1	#0
3008						XSG		

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#2 XSG A signal assigned to an X address is:
0: Fixed at the address.
1: Able to be reassigned to an arbitrary X address.

> **NOTE**
> When this parameter is set to 1, set parameter No. 3013, No. 3014, No. 3012, and No. 3019. If parameter No. 3013 and No. 3014 are not set, the deceleration signal for reference position return *DEC is assigned to bit 0 of <X0000>. If parameter No. 3012 and No. 3019 are not set, the skip signal, the PMC axis control skip signal, the measurement position arrival signal, the interlock signal for each axis direction, and the tool compensation value write signal are assigned to <X0000>.

> ⚠ **WARNING**
> Please set an appropriate value to parameters (Nos. 3013, 3014, 3012, 3019) beforehand when you set 1 to this parameter.
> If an input signal of the set address in parameters (Nos. 3013, 3014, 3012, 3019) is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used X address.

3013	X address to which the deceleration signal for reference position return is assigned

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 727

Set an address to which the deceleration signal *DECn for reference position return for each axis is to be assigned.

> **NOTE**
> This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
> Depending on the configuration of the I/O Link, the actually usable X addresses are:
> <X0000 to X0127>, <X0200 to X0327>, <X0400 to X0527>, <X0600 to X0727>

> ⚠ **WARNING**
> If you used the set address in this parameter by another usage, it may cause an unexpected machine behavior.
> Be careful enough that you must prevent the competition of the used X address.

3014	Bit position of an X address to which the deceleration signal for reference position return is assigned

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

Set a bit position to which the deceleration signal for reference position return *DECn for each axis is to be assigned.

> **NOTE**
> This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.

Alarm and message

Number	Message	Description
PS0090	REFERENCE RETURN INCOMPLETE	1. The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return. 2. An attempt was made to set the zero position for the absolute position detector by return to the reference position when it was impossible to set the zero point. Rotate the motor manually at least one turn, and set the zero position of the absolute position detector after turning the CNC and servo amplifier off and then on again.
PS0091	MANUAL REFERENCE POSITION RETURN IS NOT PERFORMED IN FEED HOLD	Manual return to the reference position cannot be performed when automatic operation is halted. Perform the manual return to the reference position when automatic operation is stopped or reset.
PS0092	ZERO RETURN CHECK (G27) ERROR	The axis specified in G27 has not returned to zero. Reprogram so that the axis returns to zero.
PS0224	ZERO RETURN NOT FINISHED	Reference position return has not been performed before the automatic operation starts. Perform reference position return only when the bit 0 (ZRNx) of parameter No. 1005 is set to 0.
PS0304	G28 IS COMMANDED WITHOUT ZERO RETURN	Although a reference position was not set, an automatic return to the reference position (G28) was commanded.
PW0001	X-ADDRESS(*DEC) IS NOT ASSIGNED.	The X address of the PMC could not be assigned correctly. This alarm may occur in the following case: - During the setting of parameter No. 3013, the X address could not be assigned correctly for the deceleration dog (*DEC) for a return to the reference position. When there are four or more paths or there are nine or more axes for one path, the signals must be assigned by setting bit 2 (XSG) of parameter No. 3008, parameter No. 3013, and parameter No. 3014.

Diagnosis data

302	Distance between the position at which the deceleration dog is turned off and the first grid point

[Data type] Real axis
[Unit of data] Machine unit

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Manual reference position return

4.2 REFERENCE POSITION SETTING WITHOUT DOG

Overview

This function moves each axis in the manual continuous feed mode near the reference position. It then sets the reference position in the reference position return mode without the deceleration signal for reference position return. This is done by setting the feed axis and direction select signal to 1. With this function, the machine reference position can be set at a given position without installing the limit switches for reference position return.

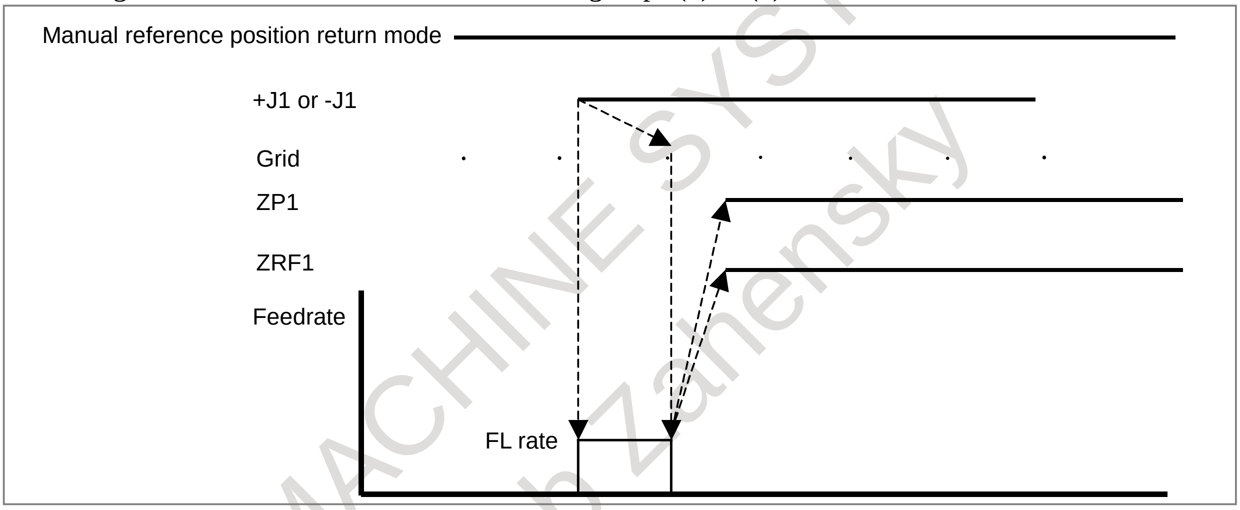
If the absolute-position detector is provided, the set reference position is retained after the power is turned off. In this case, when the power is turned on again, there is no need for setting the reference position again.

Explanation

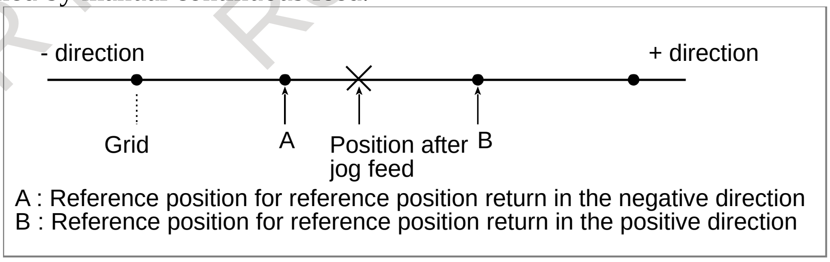
- Basic Procedure for Setting the Reference Position Without DOG

(1) Feed the tool, along the axis for which the reference position is to be set, by manual continuous feed in the reference position return direction. Stop the tool near the reference position, but do not exceed the reference position.
(2) Enter manual reference position return mode, then set 1 for the feed axis direction selection signal (for the positive or negative direction) for the axis.
(3) The CNC positions the tool to the nearest grid point (based on one-rotation signals from the position detector) in the reference position return direction specified with bit 5 (ZMIx) of parameter No.1006. The point at which the tool is positioned becomes the reference position.
(4) The CNC checks that the tool is positioned to within the in-position area, then sets the completion signal (ZPx) for reference position return and the reference position establishment signal (ZRFx) to 1.

The timing chart for the basic elements constituting steps (2) to (4) is shown below.



The following figure shows the positional relation between the reference position and the point to which the tool is positioned by manual continuous feed.



- Servo position error and one-rotation signal

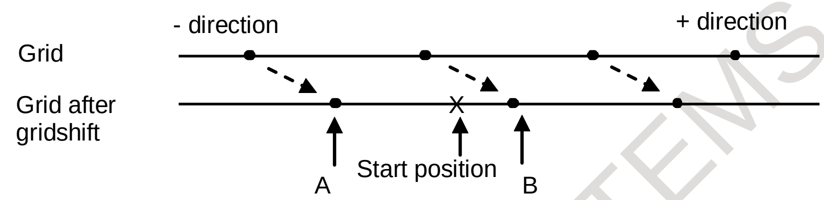
To set the reference position without DOG, when the reference position has not yet been established. The tool must be fed, in manual continuous feed mode, in the reference position return direction at such a speed that the servo position error exceeds the value set in parameter No. 1836. The tool must cross the grid line corresponding to a one-rotation signal from the position detector.
Section "MANUAL REFERENCE POSITION RETUNR" explains how to calculate the servo position error.
When the one-rotation signal from the position detector is seized at the time of reference position setting without DOG, bit 6 (PCR) of diagnosis data No. 201 is set to 1, and a grid for manual reference position return is established, enabling setting of the reference position without DOG.

> **NOTE**
> This bit has no effect until operation in the jog mode starts during setting of the reference position without DOG.

- Grid shift

To shift the reference position, the grid can be shifted by the distance set in parameter No. 1850. The grid shift to be set in the parameter must not exceed the reference counter capacity (parameter No. 1821). Reference position shift function can not be used.

When the grid has been shifted, the relationship between the start position for the reference point return operation and the reference point to be set is shown below.



A : Reference position for reference position return in the negative direction
B : Reference position for reference position return in the positive direction

- Manual reference position return

When the feed axis and direction selection signal is set to 1 in manual reference position return mode after the reference position has been established, the tool is positioned to the reference position regardless of the direction specified by the feed axis and direction selection signal. The completion signal for reference position return is then set to 1.

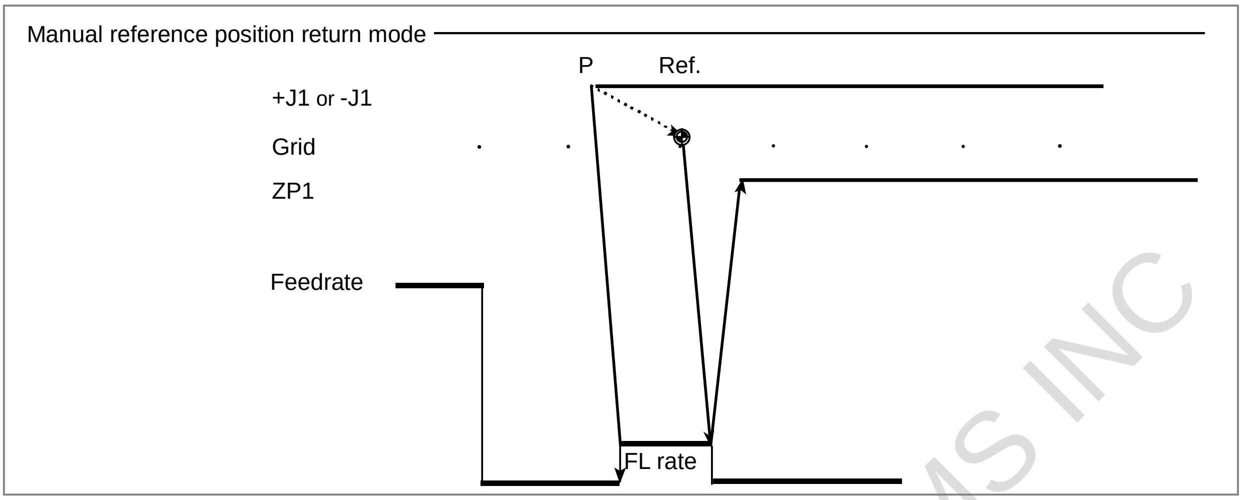
Caution

> **⚠ CAUTION**
> When a feed axis and direction selection signal is selected in the manual reference position mode, the tool may be positioned at the reference position irrespective of the direction specified by the feed axis and direction selection signal, depending on the current position.

- Setting the reference position without dog when the absolute-position detector loses the reference position

When an absolute-position detector is provided [bit 5 (APCx) of parameter No. 1815 is set to 1] and the reference position is lost [bit 4 (APZx) of parameter No. 1815 is set to 0], an alarm (DS0300) is issued.

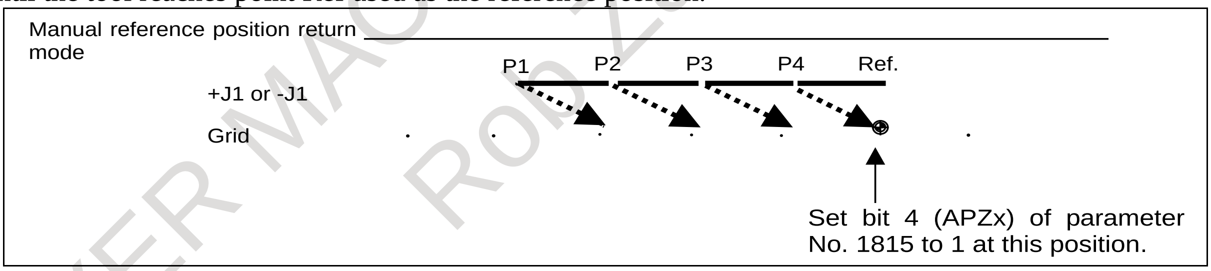
As described in "Basic Procedure for Setting the Reference Position Without DOG," feed the tool to point P in the figure below in the jog feed mode.
Then, put the machine into the manual reference position return mode and set the reference position without dog.



Upon the completion of setting the reference position without dog, the tool is positioned at a grid point and point Ref. in the figure is used as the reference position.
The correspondence between the machine position and reference position for the absolute-position detector has been established, and the reference position is also established. [Bit 4 (APZx) of parameter No. 1815 is set to 1.]
Then, reset the system to release the alarm DS0300.
With the first reference position return without dog, the correspondence between the machine position and position in the absolute-position detector has been established.

Bit 4 (GRDx) of parameter No. 1007 can be set to 1 to set the reference position without dog twice or more when an alarm DS0300 is issued due to absolute-position detection.
Bit 4 (APZx) of parameter No. 1815 remains set to 0 after the reference position is set without dog and the tool reaches the reference position.

As shown in the figure below, reference position return without dog is repeated from P2 to P3 and P4 until the tool reaches point Ref used as the reference position.



Put the system in the MDI mode at point Ref used as the reference position and set bit 4 (APZx) of parameter No. 1815 to 1.
Then, turn the power off, then on again. The correspondence between the machine position and position in the absolute-position detector has been established.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002	**IDG**							

[Input type] Parameter input
[Data type] Bit path

#7 IDG When the reference position is set without dogs, automatic setting of the bit 0 (IDGx) of parameter No. 1012 to prevent the reference position from being set again is:
0: Not performed.
1: Performed.

> **NOTE**
> When this parameter is set to 0, bit 0 (IDGx) of parameter No. 1012 is invalid.

	#7	#6	#5	#4	#3	#2	#1	#0
1005							DLZx	

[Input type] Parameter input
[Data type] Bit axis

#1 DLZx Function for setting the reference position without dogs
0: Disabled
1: Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMIx					

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#5 ZMIx The direction of manual reference position return is:
0: + direction
1: - direction

	#7	#6	#5	#4	#3	#2	#1	#0
1007				GRDx				

[Input type] Parameter input
[Data type] Bit axis

#4 GRDx When absolute position detection is performed for an axis and the correspondence between the machine position and the position on the absolute-position detector has not yet been established for the axis, reference position setting without digs is:
0: Not performed more than once.
1: Performed more than once.

	#7	#6	#5	#4	#3	#2	#1	#0
1012								IDGx

[Input type] Parameter input
[Data type] Bit axis

#0 IDGx The function for setting the reference position again, without dogs, is:
0: Not inhibited.
1: Inhibited. (The alarm PS0301 is issued.)

> **NOTE**
>
> IDGx is enabled when the bit 7 (IDG) of parameter No. 1002 is 1.
> If the function for setting the reference position without dogs is used, and the reference position is lost in absolute position detection for a cause, the alarm DS0300 is issued when the power is turned on again.
> If the operator performs reference position return, as a result of mistakenly identifying the alarm as that requesting the operator to perform a normal reference position return, an invalid reference position may be set. To prevent such an operator error, the IDGx parameter is provided to prevent the reference position from being set again without dogs.
>
> (1) If the bit 7 (IDG) of parameter No. 1002 is set to 1, the bit 0 (IDGx) of parameter No. 1012 is automatically set to 1 when the reference position is set using the function for setting the reference position without dogs. This prevents the reference position from being set again without dogs.
>
> (2) Once the reference position is prevented from being set for an axis again, without dogs, any attempt to set the reference position for the axis without dogs results in the output of an alarm PS0301.
>
> (3) When the reference position must be set again without dogs, bit 0 (IDGx) of parameter No. is set to 1012 to 0 before setting the reference position.

1250	**Coordinate system of the reference position used when automatic coordinate system setting is performed**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the coordinate system of the reference position on each axis to be used for performing automatic coordinate system setting.

1425	**FL rate of the reference position return for each axis**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

1821	Reference counter size for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

Set a reference counter size.
As a reference counter size, specify a grid interval for reference position return based on the grid method.
When a value less than 0 is set, the specification of 10000 is assumed.

1836	Servo error amount where reference position return is possible

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] 0 to 32767

This parameter sets a servo error used to enable reference position return.
In general, set this parameter to 0. (When 0 is set, 128 is assumed as the default.)
If, during reference position return, such a feedrate as exceeding a set value is not reached even once before the limit switch for deceleration is released (the deceleration signal (*DEC) for reference position return is set to 1 again), the alarm PS0090, "REFERENCE POSITION RETURN FAILURE" is issued.

1850	Grid shift and reference position shift for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -99999999 to 99999999

To shift the reference position, the grid can be shifted by the amount set in this parameter.
Up to the maximum value counted by the reference counter can be specified as the grid shift.
In case of bit 4 (SFDx) of parameter No. 1008 is 0: Grid shift
In case of bit 4 (SFDx) of parameter No. 1008 is 1: Reference point shift

> **NOTE**
> For setting the reference position without dogs, only the grid shift function can be used.
> (The reference position shift function cannot be used.)

Alarm and message

Number	Message	Description
PS0090	REFERENCE RETURN INCOMPLETE	1. The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return. 2. An attempt was made to set the zero position for the absolute position detector by return to the reference position when it was impossible to set the zero point. Rotate the motor manually at least one turn, and set the zero position of the absolute position detector after turning the CNC and servo amplifier off and then on again.
PS0091	MANUAL REFERENCE POSITION RETURN IS NOT PERFORMED IN FEED HOLD	Manual return to the reference position cannot be performed when automatic operation is halted. Perform the manual return to the reference position when automatic operation is stopped or reset.
PS0224	ZERO RETURN NOT FINISHED	Reference position return has not been performed before the automatic operation starts. Perform reference position return only when the bit 0 (ZRNx) of parameter No. 1005 is set to 0.
PS0301	RESETTING OF REFERENCE RETURN IS INHIBITED	Although bit 0 (IDGx) of parameter No. 1012 was set to 1 to inhibit the reference position from being set again for a return to the reference position without a dog, an attempt was made to perform a manual return to the reference position.
PS0302	SETTING THE REFERENCE POSITION WITHOUT DOG IS NOT PERFORMED	The reference position could not be set for a return to the reference position without a dog. Possible causes are: - The axis was not moved in the direction of a return to the reference position for jog feeding. - The axis was moved in the direction opposite to the direction of a manual return to the reference position. - Since the one-rotation signal sent from the position detector is not detected, the grid for manual reference position return is not established. (Bit 6 of diagnostic data No. 0201 must be 1.)
PS0304	G28 IS COMMANDED WITHOUT ZERO RETURN	Although a reference position was not set, an automatic return to the reference position (G28) was commanded.
DS0300	APC ALARM: NEED REF RETURN	A setting to zero position for the absolute position detector (association with reference position and the counter value of the absolute position detector) is required. Perform the return to the reference position. This alarm may occur with other alarms simultaneously. In this case, other alarms must be handled first.
DS0405	ZERO RETURN END NOT ON REF	The axis specified in automatic zero return was not at the correct zero point when positioning was completed. The position control system is in error. A return to the reference position might not be performed correctly because of an abnormality inside the CNC or servo system. Restart from a manual return to the reference position.

NOTE

For reference position setting without dog, if G28 is specified before the reference position is established, an alarm PS0304 is issued.

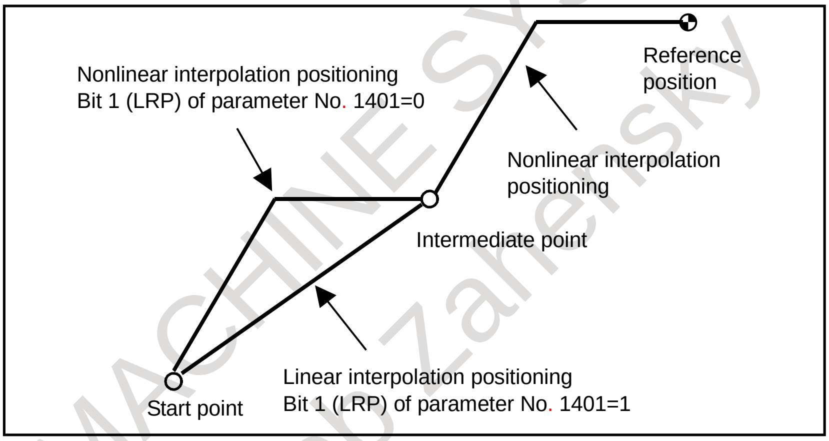
4.3 AUTOMATIC REFERENCE POSITION RETURN AND RETURN FROM THE REFERENCE POSITION

Explanation

- Automatic reference position return (G28)

The G28 command positions the tool to the reference position, via the specified intermediate point, then sets the completion signal for reference position return (see Section "Manual Reference Position Return") to 1.
The reference position must be set in parameter No. 1240 (with the coordinates specified in the machine coordinate system,).
The tool moves to the intermediate point or reference position at the rapid traverse rate.
One of the following can be selected by bit 1 (LRP) of parameter No. 1401 for the tool path to the intermediate point in automatic reference position return (G28), as with the positioning command (G00).

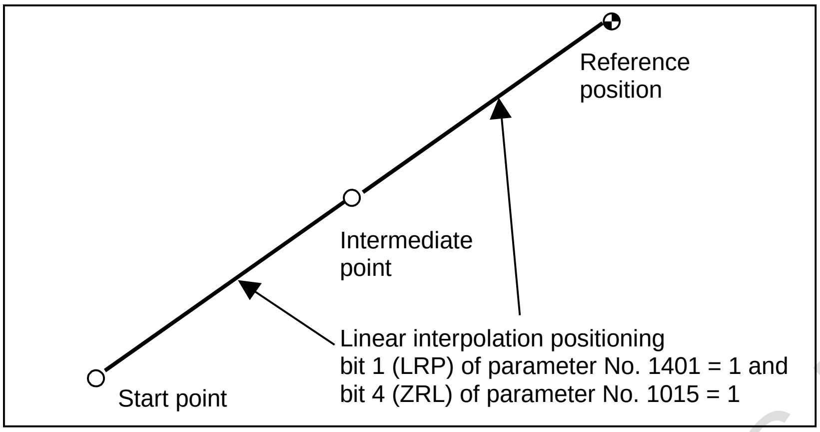
- Nonlinear interpolation positioning
  The tool is positioned with the rapid traverse rate for each axis separately. The tool path is normally straight.
- Linear interpolation positioning
  The tool is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis.



In this case, nonlinear interpolation positioning is used for the tool path from the intermediate point to the reference position.

Positioning in the machine coordinate system is used for traversing from the intermediate point to the reference position by the automatic reference position return (G28) or positioning in the machine coordinate system (G53).
Generally, nonlinear interpolation positioning is used for positioning in the machine coordinate system.

When linear interpolation positioning has been selected by setting bit 1 (LRP) of parameter No. 1401 to 1, linear interpolation positioning can be used for the tool path from the intermediate point to the reference position in automatic reference position return (G28) by setting bit 4 (ZRL) of parameter No. 1015.



> ⚠ **CAUTION**
> If G28 is specified when the reference point is not established, nonlinear interpolation positioning is used for the tool path from the intermediate point to the reference position and the same sequence as manual reference position return is used.

- Automatic reference position return (G28) if roll over is used on a rotation axis (type A)

If, in an absolute command for automatic reference position return (G28) on a rotation axis, the rotation axis is of type A (bit 0 (ROT) of parameter No. 1006 is set to 1 and bit 1 (ROS) of parameter No. 1006 is set to 0) and roll over is used (bit 0 (ROA) of parameter No. 1008 is set to 1), movement to the midpoint follows the setting of bit 1 (RAB) of parameter No. 1008, and movement from the midpoint to the reference position follows the setting of bit 5 (ZMI) of parameter No. 1006.
If, however, bit 6 (RRF) of parameter No. 1008 is set to 1, movement from the midpoint to the reference position also follows the setting of bit 1 (RAB) of parameter No. 1008.

- Restriction on automatic reference position return (G28)

There are restrictions when the following functions are used. Please refer to each function for detail.

- Mirror image
- Plane conversion
- Machine lock
- Extension of the manual handle retrace function
- Improvement of tool compensation for tool retract and recover
- Retrace
- Positioning by optimum accelerations
- In-position check disable signal
- Axis switching
- Spindle electronic gear box
- Superimposed control
- Simple spindle electronic gear box
- Electronic gear box
- Figure copying
- Programmable mirror image
- Flexible synchronization control
- Inter-path flexible synchronous control
- High precision oscillation function
- High-speed program check function
- Synchronous control
- Composite control
- Tool functions of lathe system

- Normal direction control
- Compensation function
- Coordinate system rotation
- Cutter compensation and tool nose radius compensation
- Tool length compensation

M

- Scaling

- Return from reference position (G29)

Before specifying return from the reference position (G29), return to the reference position by specifying G28 or G30.
For incremental programming, the command value specifies the incremental value from the intermediate point.
Traverse from the intermediate point and specified position is performed at the speed specified by the parameter.
When the workpiece coordinate system is changed after the tool reaches the reference position through the intermediate point by the G28 command, the intermediate point also shifts to a new coordinate system. If G29 is then commanded, the tool moves to the commanded position through the intermediate point which has been shifted to the new coordinate system. The same operations are performed also for G30 commands.

After power-up, if G29 (return from the reference position) is executed even though G28 (automatic reference position return) or G30 (2nd, 3rd or 4th reference position return) has not been executed, alarm PS0305, "INTERMEDIATE POSITION IS NOT ASSIGNED", is issued.

For the tool path in return from the reference position (G29), axis-by-axis rapid traverse rate is used for positioning to the intermediate point or the specified position. bit 1 (LRP) of parameter No. 1401 can be used to specify whether nonlinear interpolation positioning or linear interpolation positioning is used for the tool path from the reference position to the intermediate point and the tool path from the intermediate point to the specified position.

NOTE

As with the positioning command (G00), positioning in the absolute coordinate system is used for traversing from the reference position to the intermediate point or traversing from the intermediate point to the specified point. Therefore, the tool path is specified by bit 1 (LRP) of parameter No. 1401.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002					AZR			

[Input type] Parameter input
[Data type] Bit path

#3 AZR When no reference position is set, the G28 command causes:
0: Reference position return using deceleration dogs (as during manual reference position return) to be executed.
1: Alarm PS0304, "G28 IS COMMANDED WITHOUT ZERO RETURN" to be displayed.

> **NOTE**
> When reference position return without dogs is specified, (refer to bit 1 (DLZx) of parameter No. 1005) the G28 command specified before a reference position is set causes an alarm PS0304, regardless of the setting of AZR.

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMIx				ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx
#1 ROSx Setting linear or rotary axis.

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No. 1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.

ROSx	ROTx	Meaning
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

#5 ZMIx The direction of manual reference position return is:
0: + direction
1: - direction

	#7	#6	#5	#4	#3	#2	#1	#0
1007							ALZx	

[Input type] Parameter input
[Data type] Bit axis

#1 ALZx In automatic reference position return (G28):
0: Reference position return is performed by positioning (rapid traverse).
If no reference position return is performed after the power is turned on, however, reference position return is performed using the same sequence as for manual reference position return.
1: Reference position return is performed using the same sequence as for manual reference position return.

	#7	#6	#5	#4	#3	#2	#1	#0
1008		RRFx					RABx	ROAx

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROAx The rotary axis roll-over is
0: Invalid
1: Valid

NOTE
ROAx specifies the function only for a rotary axis (for which bit 0 (ROTx) of parameter No. 1006 is set to 1)

#1 RABx In the absolute programming, the axis rotates in the direction
0: In which the distance to the target is shorter.
1: Specified by the sign of command value.

NOTE
RABx is valid only when ROAx is 1.

#6 RRFx When a reference position return command (G28) is specified, the rotation direction of an absolute command for the rotary axis roll-over function follows:

0: Up to the middle point, it depends on the setting of bit 1 (RAB) of parameter No. 1008. From the middle point to the origin, it depends on the setting of bit 5 (ZMI) of parameter No. 1006.

1: It depends on the setting of bit 1 (RAB) of parameter No. 1008.

When bit 1 (ALZ) of parameter No. 1007 is 1, reference position return is performed using the same sequence as for manual reference position return.

> **NOTE**
> The setting of bit 6 (RRF) of parameter No. 1008 to 1 is valid when all of the following conditions are met:
> - Rotary axis (A type) (Bit 0 (ROT) of parameter No. 1006 is set to 1, bit 1 (ROS) of parameter No. 1006 is set to 0)
> - Roll-over is enabled (Bit 0 (ROA) of parameter No. 1008 is set to 1)
> - The reference position has been established.

	#7	#6	#5	#4	#3	#2	#1	#0
1015				ZRL				

[Input type] Parameter input
[Data type] Bit path

#4 ZRL When a reference position is established, the tool path from the middle point to the reference position and machine coordinate positioning (G53) in automatic reference position return (G28) or 2nd/3rd/4th reference position return (G30) are based on:

0: Positioning of nonlinear interpolation type

1: Positioning of linear interpolation type

> **NOTE**
> This parameter is valid when bit 1 (LRP) of parameter No. 1401 is set to 1.

1240	Coordinate value of the reference position in the machine coordinate system

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the reference position in the machine coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
1401							LRP	

[Input type] Parameter input
[Data type] Bit path

#1 LRP Positioning (G00)

0: Positioning is performed with non-linear type positioning so that the tool moves along each axis independently at rapid traverse.

1: Positioning is performed with linear interpolation so that the tool moves in a straight line.

When using 3-dimensional coordinate system conversion, set this parameter to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1614								RNO

[Input type] Parameter input
[Data type] Bit path

#0 RNO When Automatic reference position return include the Cs axis for which no reference position is established is specified,:

0: Nano interpolation of all axes is invalid.

1: Nano interpolation of all axes other than the Cs is valid.

Alarm and message

Number	Message	Description
PS0304	G28 IS COMMANDED WITHOUT ZERO RETURN	Although a reference position was not set, an automatic return to the reference position (G28) was commanded.
PS0305	INTERMEDIATE POSITION IS NOT ASSIGNED	Although a G28 (automatic return to the reference position) or G30 (return to the second, third, or fourth reference position) command was not issued after power-up, G29 (return from the reference position) was commanded.
DS0405	ZERO RETURN END NOT ON REF	The axis specified in automatic zero return was not at the correct zero point when positioning was completed. The position control system is in error. A return to the reference position might not be performed correctly because of an abnormality inside the CNC or servo system. Restart from a manual return to the reference position.

Caution

⚠ **CAUTION**

1 If G28 is specified when the reference point is not established, traversing from the intermediate point is performed in the same sequence as with manual reference position return. In this case, the direction in which the tool moves from the intermediate point is determine by the direction of reference position return set by bit 5 (ZMIx) of parameter No. 1006, so the intermediate point must be a position from which reference position return can be performed.

2 If G28 is specified when the machine is locked, the reference position return complete signal is not set to 1.

3 When data is input in millimeters for an inch machine, the reference position return complete signal may be set to 1 even when there is a difference from the reference position by the minimum setting unit programmatically. This error occurs because the least input increment is smaller than the least command increment of the machine system.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Reference position return
CONNECTION MANUAL (FUNCTION) (This manual)	Linear interpolation type G28,G30,G53

4.4 2ND REFERENCE POSITION RETURN / 3RD, 4TH REFERENCE POSITION RETURN

Overview

The G30 command positions the tool to the 2nd, 3rd, or 4th reference position, via the specified intermediate point. It then sets the completion signal for 2nd, 3rd, or 4th reference position return to 1.

The 2nd, 3rd, or 4th reference position must be set in parameter No. 1241, 1242, or 1243 with coordinates in the machine coordinate system.

On the tool path for return to the 2nd, 3rd, or 4th reference position (G30), the tool is moved to the intermediate point or reference position at the rapid traverse rate for each axis.
Bit 1 (LRP) of parameter No. 1401 can be set to specify whether to use nonlinear interpolation positioning or linear interpolation positioning for the tool path to the intermediate point during return to the 2nd, 3rd, or 4th reference position (G30) in the same way as for automatic reference position return (G28).
When bit 1 (LRP) of parameter No. 1401 is set to 1 to use linear interpolation positioning, bit 4 (ZRL) of parameter No. 1015 can be set to use linear interpolation positioning for the tool path from the intermediate point to the reference position during return to the 2nd, 3rd, or 4th reference position (G30).

Return to the 2nd, 3rd, or 4th reference position can be performed only after the reference position has been established.

- Restriction

There are restrictions when the following functions are used. Please refer to each function for detail.

- Plane conversion
- Machine lock
- Improvement of tool compensation for tool retract and recover
- Retrace
- In-position check disable signal

- Axis switching
- Superimposed control
- Electronic gear box
- Programmable mirror image
- Flexible synchronization control
- Tool functions of lathe system
- Normal direction control
- Compensation function
- Coordinate system rotation
- Cutter compensation and tool nose radius compensation
- Tool length compensation

M

- Scaling

Signal

Second reference position return end signals ZP21 to ZP28<Fn096>
Third reference position return end signals ZP31 to ZP38<Fn098>
Fourth reference position return end signals ZP41 to ZP48<Fn100>

[Classification] Output signal

[Function] The second, third, and fourth reference position end signals report the tool is at the second, third, and fourth reference positions on a controlled axis.
A numeric character appended to the end of a signal represents a controlled axis number, and a numeric character immediately following ZP represents a reference position number.

Z P x y

x : 2 ..... Second reference position return
3 ..... Third reference position return
4 ..... Fourth reference position return

y : 1 ..... Return end signal for the first axis
2 ..... Return end signal for the second axis
3 ..... Return end signal for the third axis
: :

[Output cond.] These signals turn to 1 when:

- The second, third, or fourth reference position return (G30) is completed, and the current position is in the in-position area.

These signals turn to 0 when:

- The tool moved from the reference position.
- An emergency stop is applied.
- A servo alarm is raised.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn096	ZP28	ZP27	ZP26	ZP25	ZP24	ZP23	ZP22	ZP21
Fn098	ZP38	ZP37	ZP36	ZP35	ZP34	ZP33	ZP32	ZP31
Fn100	ZP48	ZP47	ZP46	ZP45	ZP44	ZP43	ZP42	ZP41

Parameter

1241	Coordinate value of the second reference position in the machine coordinate system
1242	Coordinate value of the third reference position in the machine coordinate system

1243	Coordinate value of the fourth reference position in the machine coordinate system

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the second to fourth reference positions in the machine coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
1015				ZRL				

[Input type] Parameter input
[Data type] Bit path

#4 ZRL When a reference position is established, the tool path from the middle point to the reference position and machine coordinate positioning (G53) in automatic reference position return (G28) or 2nd/3rd/4th reference position return (G30) are based on:
0: Positioning of nonlinear interpolation type
1: Positioning of linear interpolation type

> **NOTE**
> This parameter is valid when bit 1 (LRP) of parameter No. 1401 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1401							LRP	

[Input type] Parameter input
[Data type] Bit path

#1 LRP Positioning (G00)
0: Positioning is performed with non-linear type positioning so that the tool moves along each axis independently at rapid traverse.
1: Positioning is performed with linear interpolation so that the tool moves in a straight line.
When using 3-dimensional coordinate system conversion, set this parameter to 1.

Alarm and message

Number	Message	Description
PS0046	ILLEGAL REFERENCE RETURN COMMAND	A command for a return to the second, third or fourth reference position is error. (The address P command is in error.)

Caution

⚠ **CAUTION**

1 If the G30 command is issued in machine lock status, the completion signal for 2nd, 3rd, or 4th reference position return is not set to 1.
2 If millimeter input is selected for an inch-system machine, the completion signal for 2nd, 3rd, or 4th reference position return may be set to 1, even when the programmed tool position deviates from the 2nd, 3rd, or 4th reference position by the least input increment. This is because the least input increment is smaller than the least command increment for the machine.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Reference position

4.5 IN-POSITION CHECK DISABLE REFERENCE POSITION RETURN

Overview

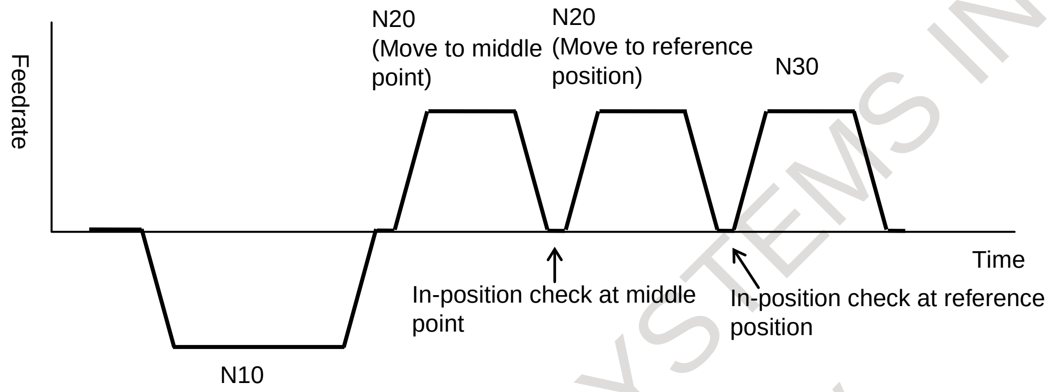
You can disable in-position check at a middle point and reference position by specifying G28.2 or G30.2 as a reference point return command.
Disabling the in-position check at the middle point and reference position reduces the cycle time.
Note that, if the G28.2 command causes reference position return at low speed, the in-position check is disabled at the middle point but enabled at the reference position.
In the operations of and restrictions on reference position return, G28.2 and G30.2 are the same as G28 and G30, except for in-position check.

Example

(1) Reference position return by G28

```
O0002;
N10 G00 X-100.0;
N20 G28 X-50.0;
N30 G00 X50.0;
N40 M30;
```

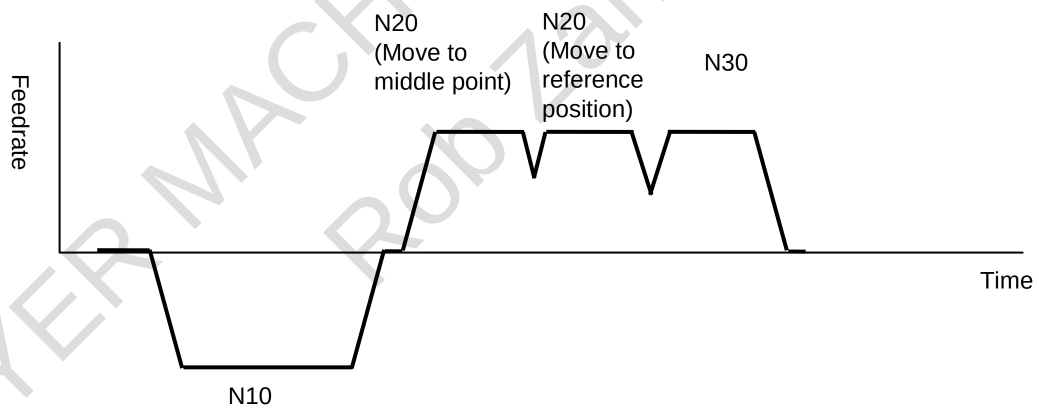
NOTE)- The program is started at coordinate X = 0.0.
- The X-coordinate of the first reference position is 0.0.



(2) Reference position return by G28.2

```
O0003;
N10 G00 X-100.0;
N20 G28.2 X-50.0;
N30 G00 X50.0;
N40 M30;
```

NOTE)- The program is started at coordinate X = 0.0.
- The X-coordinate of the first reference position is 0.0.



Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Reference position
CONNECTION MANUAL (FUNCTION) (This manual)	- Automatic reference position return and return from the reference position - 2nd reference position return / 3rd, 4th reference position return

4.6 REFERENCE POINT SETTING WITH MECHANICAL STOPPER

Overview

This function automates the procedure of butting the tool against a mechanical stopper on an axis to set a reference position. The purpose of this function is to eliminate the variations in reference position setting that arise depending on the operator, and to minimize work required to make fine adjustments after reference position setting.

Select the axis for which the reference position is to be set, then perform cycle start. Then, the following operations are performed automatically:

1. The torque (force) of the selected axis is reduced to make the butting feedrate constant, and the tool is butted against the mechanical stopper. Then, the tool is withdrawn a parameter-set distance from the mechanical stopper.
2. Again, the torque (force) of the selected axis is reduced, and the tool is butted against the mechanical stopper. Then, the tool is withdrawn a parameter-set distance from the mechanical stopper.
3. The withdrawal point on the axis is set as the reference position.

Explanation

- Basic procedure for reference point setting with mechanical stopper

(1) First, set the parameters required for reference point setting with mechanical stopper.

Bit 5 (ZMIx) of parameter No.1006...... Direction of reference position setting
Parameter No.7181................................ Withdrawal distance
Parameter No.7182................................ Reference position setting distance
Parameter No.7183................................ Butting feedrate 1
Parameter No.7184................................ Butting feedrate 2
Parameter No.7185................................ Withdrawal feedrate in reference position setting
Parameter No.7186................................ Torque limit (for 0% to 39%)
Parameter No.7187................................ Torque limit (for 0% to 100%)

(2) Select manual reference position return mode.

(3) By using a manual handle feed axis select signal (HS1A to HS1E), select the axis on which the reference position is to be set.

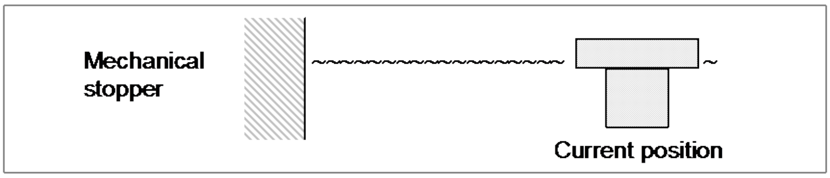
(4) Perform cycle start.

This starts the cycle operation for reference position setting.

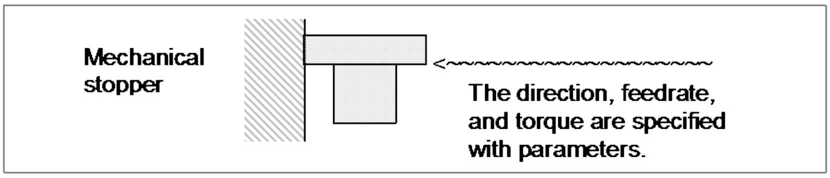
> **NOTE**
> To set the torque limit to more than 39%, set parameter No.7187.

- Cycle operation

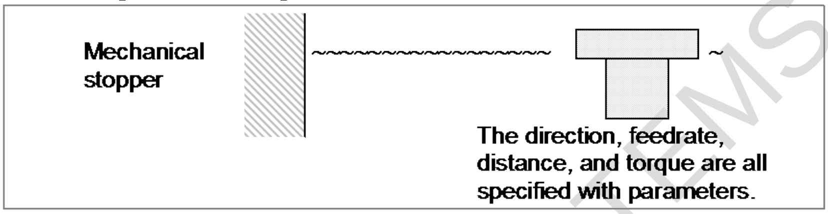
When no reference position has been set, operations (A) to (E), below, are performed automatically to set a reference position.



(A) The tool is moved along a specified axis with a limited torque until it butts against the mechanical stopper.
- The tool is moved in the direction specified with bit 5 (ZMIx) of parameter No.1006,
- At the feedrate specified with parameter No.7183,
- At the torque specified with parameter No.7186 or No.7187 (until the tool strikes the mechanical stopper).

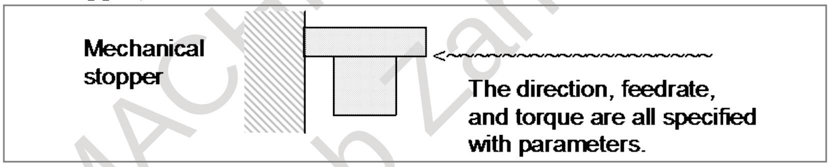


(B) After the tool strikes the mechanical stopper, the tool is withdrawn in the direction opposite to the butting direction, along the axis for a parameter-set distance.
- The tool is moved in the direction opposite to that specified with bit 5 (ZMIx) of parameter No.1006,
- At the feedrate specified with parameter No.7185,
- For the distance specified with parameter No.7181.

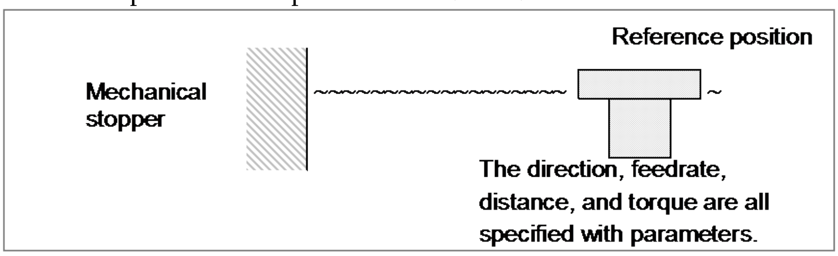


(C) Operations (D) and (E) are performed from the withdrawal point, such that the tool is butted against the mechanical stopper at a constant feedrate in reference position setting.

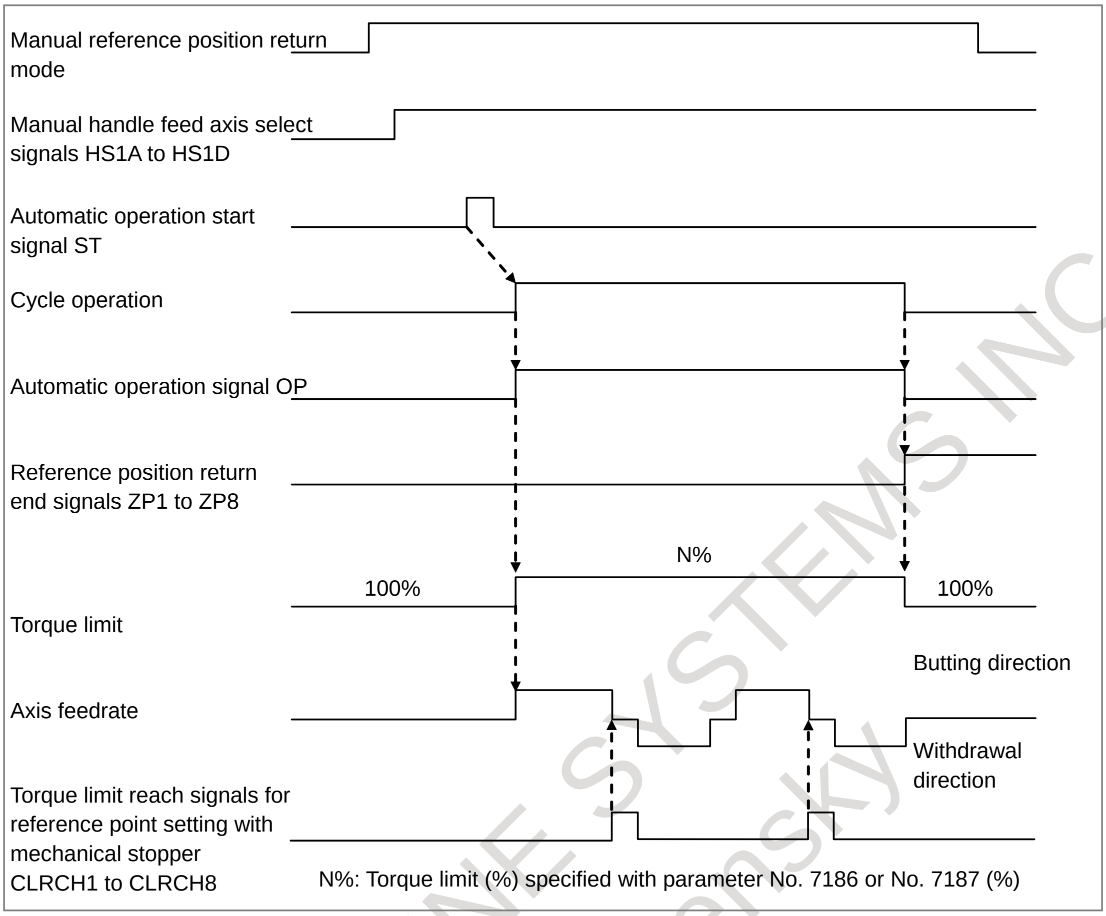
(D) The tool moves along the specified axis at a specified torque until it butts against the mechanical stopper.
- The tool moves in the direction specified with bit 5 (ZMIx) of parameter No.1006,
- At the feedrate specified with parameter No.7184,
- At the torque specified with parameter No.7186 or No.7187 (until the tool strikes the mechanical stopper).



(E) After the tool strikes the mechanical stopper end on the axis, the tool is withdrawn in the direction opposite to the butting direction, along the axis for a parameter-set distance.
- The tool is moved in the direction opposite to that specified with bit 5 (ZMIx) of parameter No.1006,
- At the feedrate specified with parameter No. 7185,
- For the distance specified with parameter No.7182.



For parameter No.7183 and No.7184, set the feedrates at which the tool is moved toward the mechanical stopper with a limited torque, considering the machine accuracy.
After the tool strikes the mechanical stopper, and the tool is withdrawn the distance specified with parameter No.7182, the withdrawal point is set as the reference position on the specified axis. Then, the reference position return end signal and reference position establishment signal are set to 1.
The timing chart for the cycle operation is shown below.



- Cycle operation after the reference position has been established

When the reference position has been established, cycle operation is not performed by operations for reference position setting with a mechanical stopper. In this case, the tool is positioned to the reference position in the rapid traverse mode. After the completion of the positioning, the relevant reference position return end signal is set to 1.
This signal is not set to 1 during automatic operation.

Signal

Cycle start signal ST<Gn007.2>

[Classification] Input signal
[Function] Starts automatic operation.
[Operation] When signal ST is set to 1 then 0 in memory (MEM) mode, DNC operation mode (RMT) or manual data input (MDI) mode, the CNC enters the cycle start state and starts operation.

MEM, RMT, or MDI mode

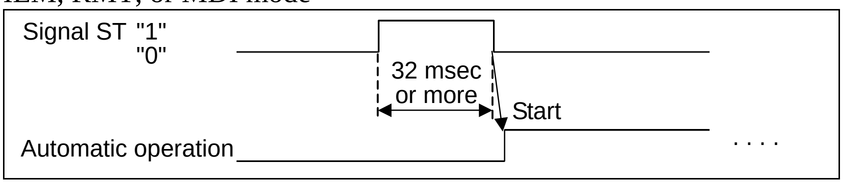


Fig.4.6 (a)

Manual Handle Feed Axis Selection Signals

HS1A to HS1D<Gn018.0 - 3>, HS1E<Gn411.0>,
HS2A to HS2D<Gn018.4 - 7>, HS2E<Gn411.1>,
HS3A to HS3D<Gn019.0 - 3>, HS3E<Gn411.2>,
HS4A to HS4D<Gn020.0 - 3>, HS4E<Gn411.3>,
HS5A to HS5D<Gn379.0 - 3>, HS5E<Gn412.0>

[Classification] Input signal

[Function] Selects the axis of manual handle feed. A set of five code signals, A, B, C, D and E is provided for each manual pulse generator. (Up to 5 generators can be used.) (For multi-path control, there is a set of code signals in each path.) The number in the signal name indicates the number of the manual pulse generator to be used.

HS1A

1 : Selection of axis to be moved by 1st. manual pulse generator
2 : Selection of axis to be moved by 2nd. manual pulse generator
3 : Selection of axis to be moved by 3rd. manual pulse generator
4 : Selection of axis to be moved by 4th. manual pulse generator
5 : Selection of axis to be moved by 5th. manual pulse generator

Code signals A, B, C, D, and E correspond to the feed axes listed in following table:

Manual handle feed axis selection signals					Feed axis
HSnE	HSnD	HSnC	HSnB	HSnA	
0	0	0	0	0	No selection (None of axis is fed)
0	0	0	0	1	1st axis
0	0	0	1	0	2nd axis
0	0	0	1	1	3rd axis
to					to
1	0	1	1	0	22nd axis
1	0	1	1	1	23rd axis
1	1	0	0	0	24th axis

Torque limit reach signals for reference point setting with mechanical stopper

CLRCH1 to CLRCH8 <Fn180>

[Classification] Output signal

[Function] These signals are used to post notification of the torque limit having been reached for each corresponding axis during cycle operation for reference point setting with mechanical stopper.

[Output cond.] Each signal is set to 1 when:

- The torque limit is reached for the corresponding axis during cycle operation for reference point setting with mechanical stopper.

Each signal is set to 0 when:

- The torque limit is not reached for the corresponding axis during cycle operation for reference point setting with mechanical stopper.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn007						ST		
Gn018	HS2D	HS2C	HS2B	HS2A	HS1D	HS1C	HS1B	HS1A
Gn019					HS3D	HS3C	HS3B	HS3A
Gn020					HS4D	HS4C	HS4B	HS4A
Gn379					HS5D	HS5C	HS5B	HS5A
Gn411					HS4E	HS3E	HS2E	HS1E
Gn412								HS5E
Fn180	CLRCH8	CLRCH7	CLRCH6	CLRCH5	CLRCH4	CLRCH3	CLRCH2	CLRCH1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMIx					

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#5 ZMIx The direction of manual reference position return is:
0: + direction
1: - direction

7181	First withdrawal distance in reference point setting with mechanical stopper
7182	Second withdrawal distance in reference point setting with mechanical stopper

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the distance, by which an axis is withdrawn after the axis butts against the mechanical stopper in each cycle operation, (the distance from the mechanical stopper to the withdrawal point).

> **NOTE**
> Set the same direction as that set in bit 5 (ZMIx) of parameter No.1006. Cycle operation cannot be started if the opposite direction is set.

7183	First butting feedrate in reference point setting with mechanical stopper
7184	Second butting feedrate in reference point setting with mechanical stopper or butting feedrate in reference point setting with mechanical stopper by Grid Method
7185	Withdrawal feedrate (common to the first and second butting operations) in reference point setting with mechanical stopper

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets a feedrate used to butt against the stopper along an axis in each cycle.

7186	Torque limit value in reference point setting with mechanical stopper

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 100

This parameter sets a torque limit value. A value from 0 to 100 corresponds to 0% to 39%. The torque limit value is obtained by multiplying the setting by 1/255. If more than 39% is to be set, use parameter No. 7187.

NOTE
When 0 is set in this parameter, 100% is assumed.

7187	Torque limit value in reference point setting with mechanical stopper or reference point setting with mechanical stopper by Grid Method

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 255

This parameter sets a torque limit value. As for the setting value, 0-255 corresponds to 0% to 100%.

When this parameter is set for reference point setting with mechanical stopper, parameter No.7186 is ignored. When this parameter is set to 0, the setting of parameter No.7186 is used.

However, in case of reference point setting with mechanical stopper by Grid Method, only this parameter is valid. If this parameter is set to 0, 100% of the rated torque is assumed even if parameter No.7186 is set up.

Alarm and message

Number	Message	Description
PS0307	CAN NOT START REFERENCE RETURN WITH MECHANICAL STOPPER SETTING	An attempt was made to set a butt-type reference position for an axis for which to use the function to set a reference position without a dog.
SV0047	REFERENCE POINT SETTING WITH MECHANICAL STOPPER IS BROKEN OFF	Reference point setting with mechanical stopper is broken off. Reference point setting again after reset.

Warning

⚠ **WARNING**

When cycle operation of Reference point setting with mechanical stopper is completed, Reference position return end signals ZPx <Fn094> turns to "1". Before completing the cycle operation, namely, when the RESET is executed or operation mode is changed to a mode other than REF mode in the state of ZPx signal is "0", the cycle operation is broken off. In such a case, take the action along the following procedure.
The situations that require interruption, it is assumed that the following situations: When the cycle operation is started the axis is stopped or is not moved while ZPx signal is "0" for the torque limit value is too low.

The countermeasure differs as follows in the case with and without an alarm SV0047, "REFERENCE POINT SETTING WITH MECHANICAL STOPPER IS BROKEN OFF" occurs during the interruption.

<u>Without alarm SV0047</u>

1 Enter state emergency stop.
2 In the case of axis that uses the absolute position detector (bit 5 (APC) of parameter No.1815 is 1), bit 4 (APZ) of parameter No.1815 is set to 0.
3 If there is a factor in stopping axis, remove it. In addition, change the parameters such as the torque limit value as needed.
4 Turn the power off, then on again.
5 Start over Reference point setting with mechanical stopper again from beginning.

<u>With alarm SV0047</u>

1 If there is a factor in stopping axis, remove it. In addition, change the parameters such as the torque limit value as needed.
2 Release alarm by reset operation.
3 Start over Reference point setting with mechanical stopper again from beginning.

Note

NOTE

1 Multiple axes cannot be selected simultaneously.
2 This cycle is not executed if either of the following conditions is satisfied:
 (A) Automatic operation signal OP <Fn000.7> is set to 1.
 (B) The direction of reference position return (bit 5 (ZMIx) of parameter No. 1006) does not agree with the sign of a reference position setting distance (parameter No. 7181 or 7182).
3 During cycle operation, excess error and feed hold in the moving and stopped states are not checked.
4 During withdrawal operation, the torque limit is effective, but the tool is not butted when the limit is reached. The torque limit is released after the reference position is established.
5 When the RESET signal is input during cycle operation, operation stops, but the torque limit is not released.

NOTE

6 This function cannot be used together with the grid shift function or reference position shift function.
7 This function cannot be used for angular, synchronous, Cs, spindle positioning, or index table indexing axis.
8 Acceleration/deceleration is applied for this function in the same way as for manual reference position return.
9 When this function is used and the manual reference position return mode is selected, feed axis and direction selection signals (such as +J1 and -J1) must be disabled.

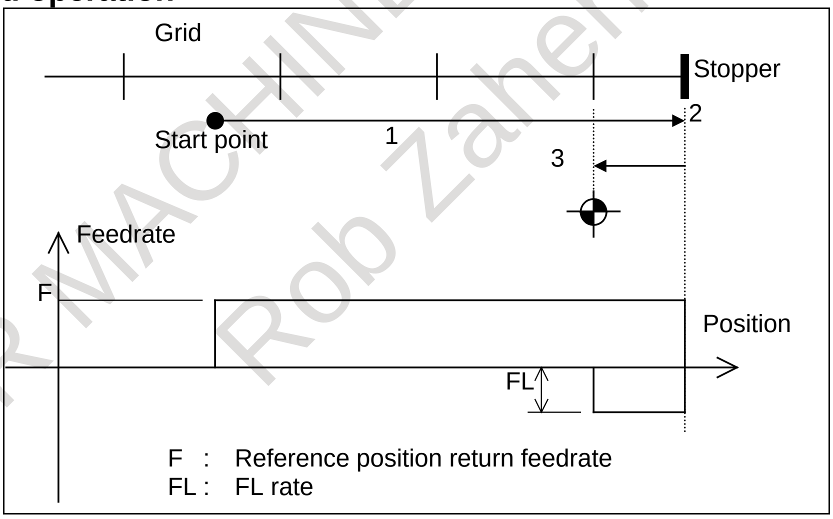
4.7 REFERENCE POINT SETTING WITH MECHANICAL STOPPER BY GRID METHOD

Overview

A reference position return for an axis can be made by pushing the axis against the mechanical stopper without using a limit switch or deceleration dog. As this reference position setting uses a grid method, its precision is on the same level as for a manual reference position return. Unlike the deceleration dog, however, the mechanical stopper cannot be shifted for reference position setting. It requires using also the "reference position shift function" or "grid shift function."

Explanation

- Setting and operation



- Setting

(1) Set a mechanical stopper near the reference position.
(2) Specify the direction of a reference position return as toward the stopper (using bit 5 (ZMIx) of parameter No. 1006).
(3) Set parameter No.7187 with the motor torque limit override value when the axis is pushed against the stopper.
(4) Set parameter No.7184 with the feedrate (reference position return feedrate) at which the axis is pushed against the stopper.
(5) Set parameter No.1425 with the feedrate (reference position return FL rate) at which the axis moves to the reference position from the stopper.

- Operation

(1) The machine is placed in the reference position return mode, and the feed axis direction select signal is issued to feed the axis toward the stopper. At this time, torque limit override is put in effect.
(2) When the axis hits the stopper, it stops because of a torque limit. When it is detected that the axis has stopped because of a torque limit, the axis starts moving back.
(3) When the axis stops at the first grid, its reference position return is completed. However, grids are ignored if a parameter specifies so, until the amount of servo position deviation changes from positive to negative or vice versa.

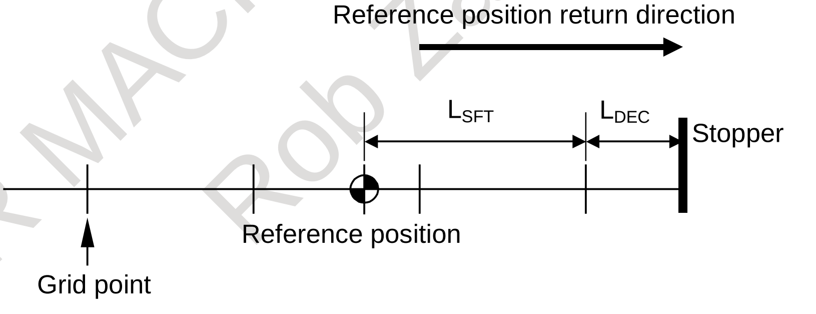
> **NOTE**
> An alarm (PS0090), "REFERENCE RETURN INCOMPLETE", is raised if an attempt is made to perform reference point setting with mechanical stopper by Grid Method without establishing electrical grids based on the one-rotation signal from the position detector.

Reference position adjustment by grid shifting

The reference position can be shifted by as much as an amount of grid position shift specified in parameter No.1850.
A maximum grid shift amount that can be specified is below a reference counter capacity specified in parameter No.1821 (grid interval).

Reference position adjustment by reference position shifting

The reference position can be shifted by specifying an amount of reference position shift in parameter No.1850.
This function is enabled by setting the bit 4 (SFDx) of parameter No. 1008 to 1.
The reference position can be shifted by specifying the reference position shift amount shown below in parameter No.1850.
The distance L_{DEC} the axis moved for a reference position return is automatically saved to parameter No.1844.



L_{DEC} : Distance the axis moves to get to the first grid point from the stopper if the reference position shift amount is 0 (parameter No.1844).
L_{SFT} : Reference position shift amount (parameter No.1850)

- Procedure

(1) Set up the following parameters. Then, turn the power off.
Bit 4 (SFDx) of parameter No. 1008 is set to 1.
Parameter No.1844 is set to 0.
Parameter No.1850 is set to 0.
(2) Make a reference position return. The machine first hits the stopper and then stops at the first grid point. The distance L_{DEC} is automatically saved to parameter No.1844. Do not change the automatically set value.
(3) Obtain the distance (reference position shift amount L_{SFT}) from the stop position to the reference position, and set it in parameter No.1850. Then, turn the power off.

This is the end of reference position adjustment.

(4) The machine will stop at the reference position from now on whenever a reference position return is made.

> **NOTE**
> 1 When reference position return with mechanical stopper by grid method is performed while reference position is established, reference position return with mechanical stopper by grid method is performed again.
> 2 When the relation of the sign of parameters Nos.1844 and 1850 is contradicted, the amount of the grid shift becomes effective only below the capacity of the Reference counter.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMIx					

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#5 ZMIx The direction of manual reference position return is:
0: + direction
1: - direction

	#7	#6	#5	#4	#3	#2	#1	#0
1008				SFDx				

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#4 SFDx In reference position return based on the grid method, the reference position shift function is:
0: Disabled
1: Enabled

1425	FL rate of the reference position return for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

1821	Reference counter size for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

Set a reference counter size.
As a reference counter size, specify a grid interval for reference position return based on the grid method.
When a value less than 0 is set, the specification of 10000 is assumed.
When a linear scale with absolute address reference marks is used, set the interval of mark 1.

1844	Distance to the first grid point when the reference position shift amount in the reference position shift function is 0 or when a reference position return is made by grid shift

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999999999 to 999999999

(1) When the reference position shift function is enabled (when bit 4 (SFDx) of parameter No. 1008 is set to 1)
Set the distance (detection unit) to the first grid point from a point at which the deceleration dog is released when the reference position shift (parameter No. 1850) is set to 0.
(2) When a reference position return is made by grid shift with a setting not to use reference position setting without dogs (when bit 4 (SFDx) of parameter No. 1008 is set to 0, and bit 1 (DLZx) of parameter No. 1005 is set to 0)
Set the distance to the first grid point from a point at which the deceleration dog is released. (Detection unit)
(3) When a reference position return is made by grid shift with a setting to use reference position setting without dogs (when bit 4 (SFDx) of parameter No. 1008 is set to 0, and bit 1 (DLZx) of parameter No. 1005 is set to 1)
Set the distance from the start position for reference position setting without dogs to the first grid point. (Detection unit)

> **NOTE**
> 1 When the reference position shift function is enabled (when bit 4 (SFDx) of parameter No. 1008 is set to 1)
> When bit 4 (SFDx) of parameter No. 1008 is set to 1, the distance from a point at which the deceleration dog is released to the first grid point (parameter No. 1844) is set to 0, and reference position shift (parameter No. 1850) is set to 0, a manual reference position return allows this parameter to be set automatically. Do not change an automatically set value.

> **NOTE**
> 2 When a reference position return is made by grid shift with a setting not to use reference position setting without dogs (when bit 4 (SFDx) of parameter No. 1008 is set to 0, and bit 1 (DLZx) of parameter No. 1005 is set to 0)
> When a manual reference position return using deceleration dogs is made, this parameter is set automatically.
> 3 When a reference position return is made by grid shift with a setting to use reference position setting without dogs (when bit 4 (SFDx) of parameter No. 1008 is set to 0, and bit 1 (DLZx) of parameter No. 1005 is set to 1)
> When a reference position setting without dogs is made, this parameter is set automatically.

1850	Grid shift and reference position shift for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -99999999 to 99999999

To shift the reference position, the grid can be shifted by the amount set in this parameter. Grid shift amount that can be set is a value less than reference counter size.
When bit 4 (SFDx) of parameter No.1008 is 0, it is grid shift, and when bit 4 (SFDx) of parameter No.1008 is 1, it is reference point shift.

> **NOTE**
> For setting the reference position without dogs, only the grid shift function can be used. (The reference position shift function cannot be used.)

7184	Second butting feedrate in reference point setting with mechanical stopper or butting feedrate in reference point setting with mechanical stopper by Grid Method

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets a feedrate used to butt against the stopper along an axis.

7186	Torque limit value in reference point setting with mechanical stopper

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 100

This parameter sets a torque limit value. A value from 0 to 100 corresponds to 0% to 39%. The torque limit value is obtained by multiplying the setting by 1/255. If more than 39% is to be set, use parameter No. 7187.

> **NOTE**
> When 0 is set in this parameter, 100% is assumed.

7187	Torque limit value in reference point setting with mechanical stopper or reference point setting with mechanical stopper by Grid Method

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 255

This parameter sets a torque limit value. As for the setting value, 0-255 corresponds to 0% to 100%.
When this parameter is set for reference point setting with mechanical stopper, parameter No.7186 is ignored. When this parameter is set to 0, the setting of parameter No.7186 is used.
However, in case of reference point setting with mechanical stopper by Grid Method, only this parameter is valid. If this parameter is set to 0, 100% of the rated torque is assumed even if parameter No.7186 is set up.

	#7	#6	#5	#4	#3	#2	#1	#0
7188	RNWx							GRSx

[Input type] Parameter input
[Data type] Bit axis

#0 GRSx Reference point setting with mechanical stopper by Grid Method is:
0: Not performed.
1: Performed.

#7 RNWx During reference point setting with mechanical stopper by Grid Method, until the sign of servo position deviation is inverted, the grid is
0: Not ignored.
1: Ignored.

Warning

> ⚠ **WARNING**
> When cycle operation of Reference point setting with mechanical stopper by grid method is completed, Reference position return end signals ZPx <Fn094> turns to "1".
> Before completing the cycle operation, namely, when the RESET is executed or operation mode is changed to a mode other than REF mode in the state of ZPx signal is "0", the cycle operation is broken off. In such a case, emergency stop is applied. And, start over Reference point setting with mechanical stopper by grid method after emergency stop cancellation.

4.8 DISTANCE CODED LINEAR SCALE INTERFACE

Overview

The interval of each reference marks (one-rotation signal) of distance coded linear scale are variable. Accordingly, if the interval is determined, the absolute position can be determined. The CNC measures the interval of reference marks by axis moving of short distance and determines the absolute position. Consequently the reference position can be established without moving to reference position.

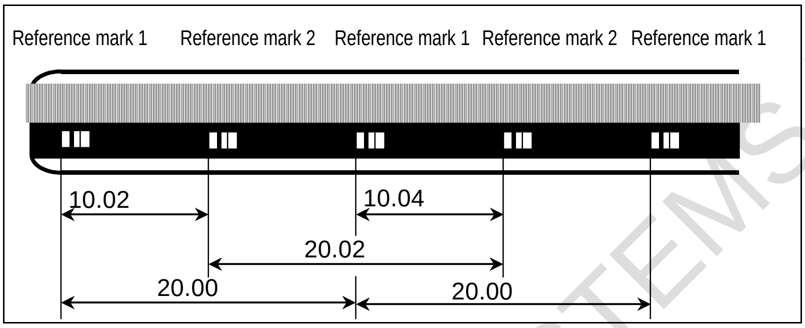


Fig. 4.8 (a) Example of distance coded linear scale

Procedure for reference position establishment

- Procedure

(1) Select the JOG mode, and set the manual reference position return selection signal ZRN to 1.
(2) Set the feed axis and direction selection signal(+J1, -J1, +J2, -J2, …) to "1" and feed the axis for which you want to establish a reference point.
(3) The axis is fed at a constant low speed (reference position return FL feedrate specified by parameter No. 1425 setting).
(4) When a reference mark is detected, the axis stops, then the axis is fed at a constant low speed again.
(5) Above (4) is executed repeatedly until two, three or four reference marks are detected. And absolute position is determined and the reference position establishment signal (ZRF1, ZRF2, ZRF3, …) turns to 1.
(A number of reference marks is determined by the bits 2 (DC2) and 1 (DC4x) of parameter No. 1802.)

> ⚠ **CAUTION**
> When the scale correctly detects the mark, the reference position is correctly established regardless of the number of detected reference marks. When the scale misreads the mark position, the alarm DS1449 "REFERENCE MARK ARE DIFFERENT FROM PARAMETER" is usually issued. However, if the interval of the detected mark is same as the parameters, the alarm might not be issued and the reference position might be established with the wrong position.
> It is reliable to avoid misreading the reference marks by using more number of detecting reference marks. It is recommended that the reference position is established by detecting four reference marks though the number of detecting reference marks can be selected from two to four reference marks number.

The timing chart for this procedure is given below.

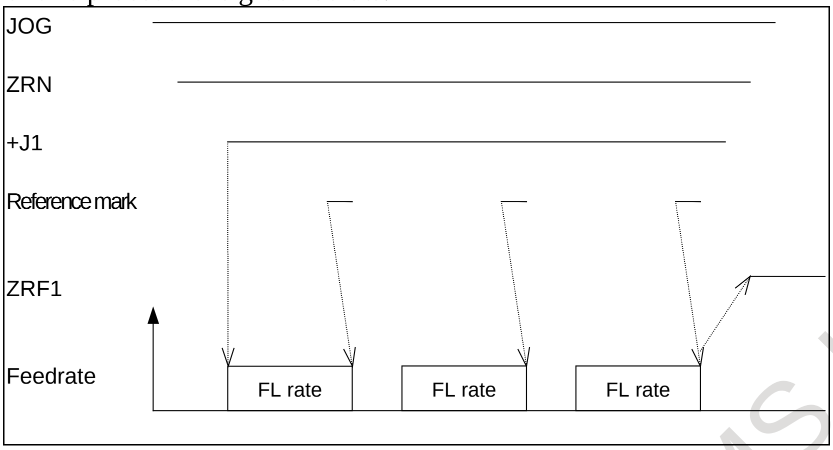


Fig. 4.8 (b) Timing chart for reference position establishment

- Procedure for establishing a reference position through automatic operation

If an automatic reference position returns (G28) is specified before a reference position is not established, steps (3) to (5) above are performed automatically.
After the reference position is established, the automatic reference position return is performed.

- Stopping the operation for establishing a reference position

The operation for establishing a reference position is stopped if any of the following operations is performed in steps (3) to (5), described above.

- Reset
- Setting the feed axis and direction selection signal (+J1, -J1, +J2, -J2, etc.) to 0

If any of the following operations is performed during the operation of automatic reference position return (G28) before a reference position is not established, the operation for establishing a reference position stops:

- Reset
- Performing feed hold during movement from an intermediate position

If the operation for establishing a reference position is stopped by an operation other than a reset, the operation for establishing a reference position must be reset and resumed.

Explanation

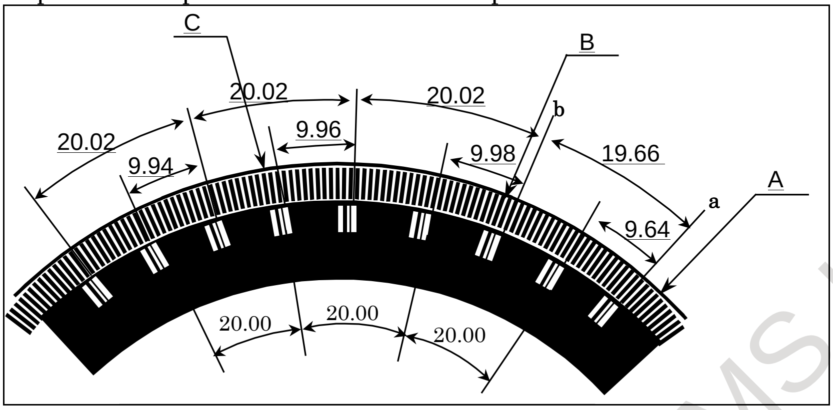
- Reference Position Return

(1) When axis is moved in REF mode while reference position is not established, reference position establishment procedure is executed.
(2) When axis is moved in REF mode while reference position is established, reference position establishment procedure is not executed and axis is moved to reference position.
(3) When the reference position is not established and the reference position return command (G28) is executed, the reference position establishment procedure is executed. The next movement the axis depends on the setting of bit 0 (RFS) of parameter No. 1818.
(4) When the reference position is already established and the reference position command (G28) is executed, the movement of the axis depends on the setting of bit 1 (RF2) of parameter No. 1818.

- Distance coded rotary encoder

In case of setting a rotary axis, if bit 3 (DCRx) of parameter No. 1815 is set, the setting axis is regarded as being equipped with a distance coded rotary encoder.
In case of distance coded rotary encoder, the marker interval may be different from parameter setting value at the joint of the encircled circle (a-b section of the following figure). When the reference point return is executed through this section, it is not able to establish the reference point. Therefore, in case of

distance coded rotary encoder, if the reference point return is started for B point from A point of below figure, the reference point is not established yet at B point. The reference point return is re-started for C point. The reference point return procedure is finished at C point.



- In case of distance coded rotary encoder, only the measurement by three points or four points is possible. (Bit 2 (DC2) of parameter No. 1802 is disregarded as 0.)

- Rotary axis (B type)

The distance coded rotary encoder does not have the rotary data. For example, 30.000 degree and 390.000 degree have completely same scale data though their position is different from each other by just one rotation. When distance coded rotary encoder is used and rotary axis is B type (bit 0 of parameter No.1006 is 1 and bit 1 of parameter No.1006 is 1 (machine coordinate system of rotary axis is linear axis type)), reference position established by this function is rounded by the amount of movement per a rotation of rotary axis even if the axis is rotated more than one rotation.
For this case, when the Reference point is established, correct machine coordinates can be established even the turn of the machine more than one revolution by setting the bit 0 (RVSx) of parameter No.1815 to "1".
Moreover, the present machine coordinates are stored immediately before the power is turned off when the parameter RSVx is set to 1. Therefore, immediately after the power is turned on again, this function can restore the approximate machine coordinates and enables the function for checking the stored stroke during the time from power-on to the reference position establishment. Even before the reference position is established by reference position return, the stored stroke check can be performed using approximate machine coordinates. Absolute coordinates are set based on the machine coordinates. Data such as workpiece offsets specified by G92, G52, and so on before the power is turned off, however, is not set. If bit 3 (PPD) of parameter No. 3104 is set to 1, the relative position indication is preset.

Axis synchronous control

In case of the synchronization establishment of the axis synchronous control, when the position is set immediately after turning on of the power, the synchronization establishment is executed.

Parameter setting procedure

1 The bit 0 (RVSx) of parameter No.1815 is set to 0, the power supply of CNC is turned off once, and it turns on again.
2 The reference position is established by manual reference position return.
3 The reference position returns, and it moves to the reference position.
4 The bit 0 (RVSx) of parameter No.1815 is set to 1, the power supply of CNC is turned off once, and it turns on again.

NOTE

1. a machine coordinate value may get out over a rotation, in the case of moving over 180 degree during turning off, because CNC saves a machine coordinate value just before CNC turns off and in following turning on get from the value.
2. Temporary absolute coordinate setting and this function cannot be used together within same axis.
3. This function can be used only when the amount of one rotation of rotary axis is 360 degree. Set 0 to the parameter No.1869.
   If the values other than 0 are set, an alarm DS1448, "ILLEGAL PARAMETER (D.C.S.)", is issued.
4. Set the parameter No.1240 to 0 because the reference position must be 0 degree.
5. If the tool is already beyond a stored stroke immediately after the power is turned on, an overtravel alarm (OT alarm) is issued.
   In this case, release the alarm by moving the tool along the axis in the direction that does not cause overtravel.
   If bit 6 (LZR) of parameter No. 1300 is set to 1, the function for checking the stored stroke during the time from power–on to the reference position establishment is disabled.
6. When the power is turned on while the CAN key and the <P> address key are held down, the stored stroke check immediately after power-on is disabled.
   (When the MDI keys are provided as standard keys)
   (When performing this operation, exercise special care.)
   In this case, machine coordinates and absolute coordinates are set.
   When the compact type MDI keys are used, the stored stroke immediately after power-on is disabled by holding down the CAN key and <O> address key at power-on.

- Requirements when this function is used with axis synchronous control axes

When this function is used with axis synchronous control axes, the distance coded linear scale used for the master axis and that used for the slave axis must have reference marks placed at identical intervals.
(Set identical values in parameters Nos. 1821 and 1882 for both the master and slave axes.)
A distance coded linear scale can be installed on the master and slave axes either in the same or opposite direction.
This function does not work unless the use of this function is specified for both the master and slave axes (bit 2 (DCL) of parameter No. 1815 is 1).
Also, in all parameters related to this function, except parameters Nos. 1883 and 1884 (distance from the scale zero point to reference position 1, 2), set identical values for both the master and slave axes.
If a parameter value for the master axis differs from the corresponding parameter value for the slave axis, alarm SV1051 "ILLEGAL SYNCHRONOUS AXIS" is issued.

NOTE

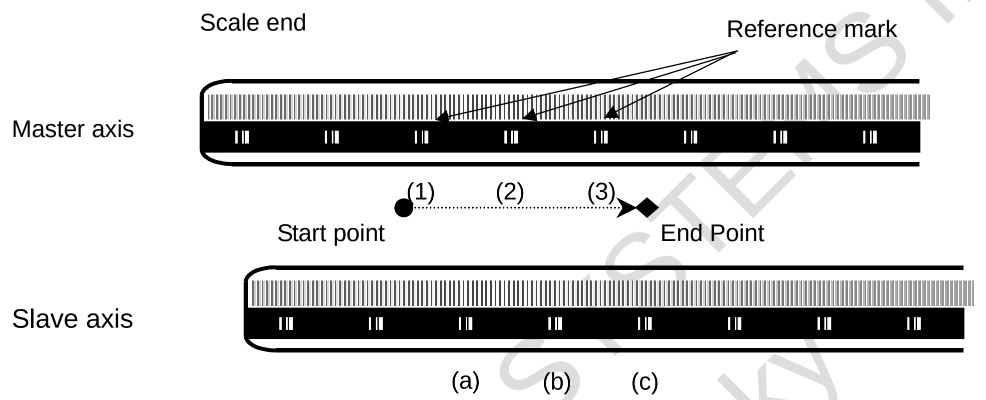
1. When this function is used with axis synchronous control axes for which the operation mode is switched between synchronization operation and normal operation, this function is enabled only if the synchronization select signal (SYNC1, SYNC2, ...) is 1. (During establishment of a reference position, the synchronization select signal status must be maintained.)
2. When this function is used by the axis synchronous control, only one axis is available for the slave axis. When the slave axis of the axis synchronous control is two or more settings, the alarm DS0027 "MISMATCH OF SYNCHRONOUS AXIS(D.C.S)" is issued.

- Reference position establishment with axis synchronous control axes

With axis synchronous control axes, a reference position is established as follows. When a reference mark for the master or slave axis is detected, a stop takes place temporarily. Then, a feed operation is performed again at the reference position return FL feedrate. This sequence is repeated until a reference mark is detected for both the master and slave axes. Then the absolute position is calculated for both the master and slave axes, and the reference position establishment signal (ZRF1, ZRF2, ...) are set to 1.

After the reference position has been established by the above operation, a synchronization error is corrected. (Checking for excessive synchronization error alarm 2 is made even during reference position establishment.)

(Example of 3 points measurement system)



In the above example, the following sequence is executed.

a. When the reference mark (1) of the master axis is detected, both master axis and slave axis stop.
b. Both the axes begin to move again at a reference position return FL feedrate.
c. When the reference mark (a) of the slave axis is detected, both axes stop again.
d. Both the axes begin to move again at FL feedrate.
e. Both axes repeat the operation until all point ((2) -> (b) ->(3)) are detected.
f. When the slave axis detects the third reference mark (c), both the axes end the reference position establishment.

> **NOTE**
> In case of this function is used with axis synchronous control axes, if the value of parameters Nos. 1883 and 1884 (Distance 1, 2 from the scale zero point to reference position) for both the master and slave axes is 0, the reference position is not established. Also, the reference position establishment signals (ZRF1, ZRF2, ...) are not set to "1".

- PMC control axis

In PMC axis control, if the reference position return command (axis control command code 05H) is issued for an axis having a distance coded linear scale, reference position return is performed according to the reference position return sequence for the distance coded linear scale.

Specifically, the following operations take place:

Before reference position establishment	The reference position is established by detecting two, three or four reference marks. Movement to the reference position is not performed.
After reference position establishment	Positioning at the reference position is performed.

- Arbitrary angular axis control

There are the following limitations when the angular axis control is used.

(a) It is necessary to use the linear scale with the distance coded reference mark for both the perpendicular axis and the angular axis.

(b) When the reference point of the perpendicular axis is established, it is necessary to establish the reference point of the angular axis previously. When perpendicular axis is commanded first or perpendicular axis and angular axis are commanded simultaneously, the alarm DS0020, "REFERENCE RETURN INCOMPLETE", is generated.

(c) During the reference position establishment operation of the angular axis, the command in the perpendicular axis is invalid in the manual reference point return.

Note

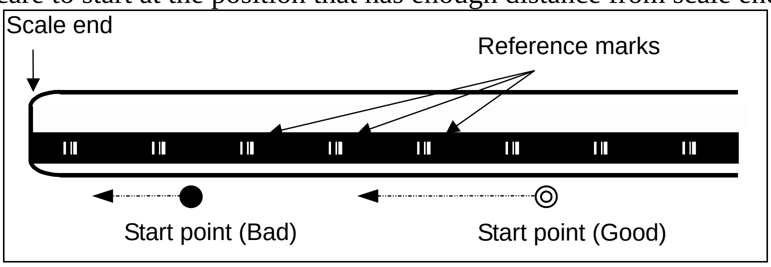
(1) In the case of the actual interval of reference marks is different from parameter setting value, the alarm DS1449, "REFERENCE MARK ARE DIFFERENT FROM PARAMETER", occurs.

(2) This function is disabled if any of the following conditions is satisfied:
- Either parameter 1821 (mark-1 interval) or parameter 1882 (mark-2 interval) is set to 0.
- Parameters Nos. 1821 and 1882 have identical settings.
- The difference between the settings made for parameters Nos. 1821 and 1882 is greater than or equal to twice either setting.
- The absolute-position detection function is enabled (bit 5 (APCx) of parameter No.1815 is 1).

(3) A difference of parameter No.1821 (mark-1 interval) and No.1882 (mark-2 interval) must be more than 4.
Example)
When the scale, which is that mark1 interval is 20.000mm and mark2 interval is 20.004mm, is used on IS-B machine:
When the detection unit of 0.001mm is selected, parameter No.1821 and No.1882 must be set 20000 and 20004, and the difference of them is 4.
To use such a scale, please adjust the detection unit by modification of parameter No. 1820 (CMR) and No.2084/2085 (flexible feed gear) to make the difference of No.1821 and 1882 more than 4 as following examples.
(a) Set the detection unit=0.0001mm, and parameter No.1821 is set to 200000, parameter No.1882 is set to 200040
(b) Set the detection unit=0.0005mm, and parameter No.1821 is set to 40000, parameter No.1882 is set to 40008

NOTE
When the detection unit is changed, parameters relating to the detection unit (such as the effective area and positional deviation limit) must also be changed accordingly.

(4) In this procedure, the axis does not stop until two, three or four reference marks are detected. If this procedure is started at the position near the scale end, the axis does not stop until over travel alarm occurs. Please care to start at the position that has enough distance from scale end.



(5) When the axis used this function, the following function can not be used.
- Absolute position detection

(6) If axial movement is made in the direction opposite to that of reference position return, the movement is reversed to the direction of reference position return after three or four reference marks

have been detected. Steps 3 to 5 of the basic procedure for establishing a reference position are carried out to establish the reference position.

(7) Straightness compensation function
When the reference position establishment of moving axis is executed after the establishment of compensation axis, the compensation axis is moved by straightness compensation amount when the reference point of moving axis is established.

(8) The reference position establishment is not performed during synchronous control is activated.

(9) The reference position establishment is not performed during composite control is activated.

(10) The reference position establishment is not performed during superimposed control is activated.

(11) No reference position can be established during flexible synchronization control.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1802						DC2x	DC4x	

[Input type] Parameter input
[Data type] Bit axis

#1 DC4x When the reference position is established on the linear scale with reference marks:
0: An absolute position is established by detecting three reference marks.
1: An absolute position is established by detecting four reference marks.

#2 DC2x Reference position establishment operation for a linear scale with reference marks is performed as follows:
0: The setting of bit 1 (DC4) of parameter No. 1802 is followed.
1: An absolute position is established by detecting two reference marks.

NOTE
1 When this parameter is set to 1, specify the direction of the scale zero point by setting bit 4 (SCP) of parameter No. 1817.
2 When a rotary encoder with absolute address reference marks is used, this parameter is invalid. Even when this parameter is set to 1, the setting of bit 1 (DC4) of parameter No. 1802 is followed.

	#7	#6	#5	#4	#3	#2	#1	#0
1815					DCRx	DCLx	OPTx	RVSx

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 RVSx When the scale without rotary data is used:
0: The CNC does not maintain the rotary data.
1: The CNC maintains the rotary data.
This parameter is effective in the axis that satisfies either of the following conditions.
- The axis is rotary axis B type and the movable range is more than one rotation.
- Bit 6 (RVLx) of parameter No.11802 is 1.

NOTE

1 In the case of a rotary axis B type whose movable range is over one rotation, a rotary scale with rotary data had better be used. Otherwise, a machine coordinate value may get out over a rotation, in the case of moving over 180 degree during turning off.

2 This parameter is available for the axis that is rotary axis B type or the axis whose bit 6 (RVLx) of parameter No.11802 is 1. This parameter is available for the axis with an absolute position detector (absolute Pulsecoder), a distance coded rotary scale interface (phase A/B) or a rotary scale with distance-coded reference marks (serial).

3 If this parameter is available, the machine coordinate value just before CNC turns off is saved. In the case of moving over the following amounts during turning off or before reference position return, a machine coordinate value may get out over a rotation because CNC saves a machine coordinate value just before CNC turns off and in following turning on get from the value.
In case of the rotary axis B type :
    Over 180 degree
In case of the linear axis type :
    Over 1/4 of values set to parameter No.11810.

4 If this parameter is changed, the correspondence between the machine position and the absolute position detector is lost. So, establish the reference position again. (Bit 4 (APZx) of parameter No.1815 is set to 0, and an alarm DS0300, “APC ALARM: NEED REF RETURN” is issued. The cause that sets bit 4 (APZx) of parameter No.1815 to 0 is indicated in bit 0 of diagnosis data No.310).

5 Absolute coordinate value is set by machine coordinate value. However, after CNC turns on, the workpiece offset such as G92 and G52 executed before CNC turns off is not set.

6 This function cannot be used together with the bit 3 (SCRx) of parameter No. 1817 that convert scale data.

7 In case of the rotary axis B type and the amount of one rotation of rotary axis is 360, set the parameter No.1869 to 0. Moreover, set the parameter No.1240 to 0 because the reference position must be 0 degree.

8 In case of the rotary axis B type and the amount of one rotation of rotary axis is not 360, set the parameter No.1869 to the amount of one rotation. Moreover, set the parameter No.1240 to 0 because the reference position must be 0 degree.

9 In case of the linear axis type, set the amount of the movement per one motor rotation to the parameter No.11810.

10 This parameter cannot be used together with the stored stroke limit check immediately after power-on (bit 0 (DOTx) of parameter No.1311 is 1).

#1 OPTx The separate position detector is:

0: Not to be used (semi-closed system)

1: To be used (full-closed system)

> **NOTE**
> Set 1 when this function is used.

#2 DCLx As a separate position detector, a linear scale with reference marks or a linear scale with an absolute address zero point is:
0: Not used.
1: Used.

> **NOTE**
> Set 1 when this function is used.

#3 DCRx As a scale with absolute address reference marks:
0: A rotary encoder with absolute address reference marks is not used.
1: A rotary encoder with absolute address reference marks is used.

> **NOTE**
> When using a rotary encoder with absolute address reference marks, bit 2 (DCLx) of parameter No. 1815 set also to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1817				SCPx				

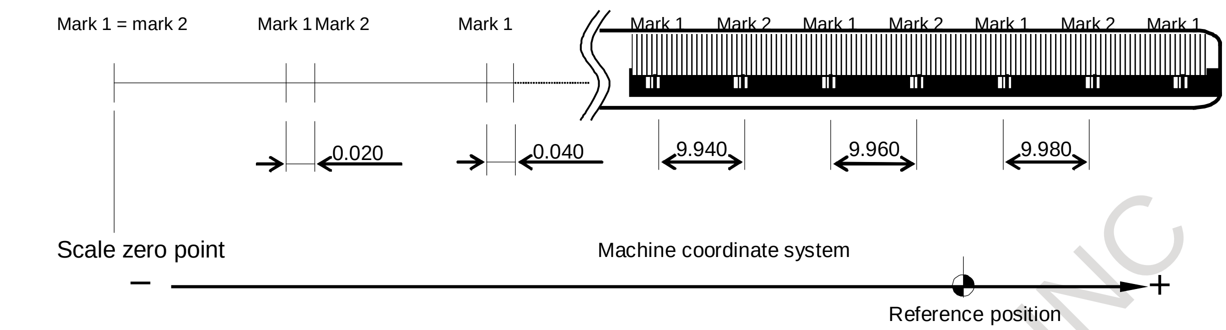
[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

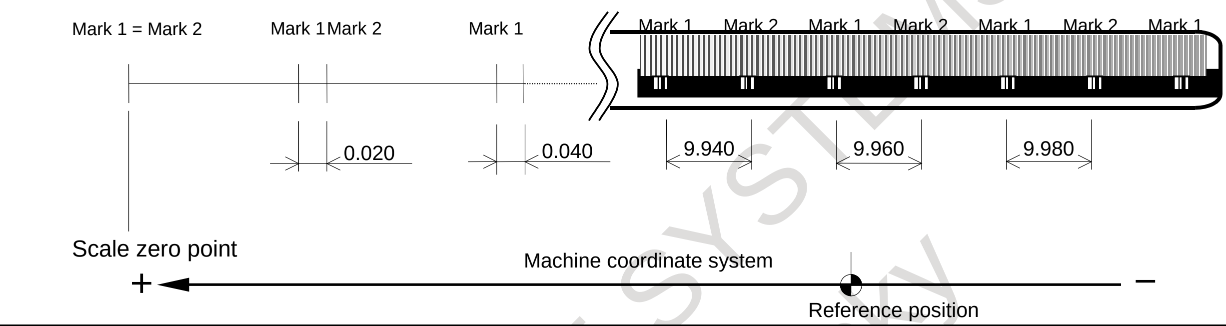
#4 SCPx For two-point measurement (when bit 2 (DC2) of parameter No. 1802 is set to 1), the scale zero point direction is:
0: On the minus side. (The reference position is located in the plus direction when viewed from the scale zero point.)
1: On the plus side. (The reference position is located in the minus direction when viewed from the scale zero point.)

> **NOTE**
> 1 This parameter is valid when bit 2 (DC2) of parameter No. 1802 is set to 1.
> 2 If this parameter is set to an incorrect value, an incorrect coordinate system is established. In such a case, reverse the setting then perform reference position establishment operation again.

- When parameter SCP = 0



- When parameter SCP = 1



	#7	#6	#5	#4	#3	#2	#1	#0
1818							**RF2x**	**RFSx**

[Input type] Parameter input
[Data type] Bit axis

#0 RFSx If G28 is specified for an axis for which a reference position is not established (ZRF = 0) when a linear scale with an absolute address zero point or a linear scale with absolute address reference marks is used:
0: A movement is made to the reference position after reference position establishment operation.
1: No movement is made after reference position establishment operation, but the operation is completed.

> **NOTE**
> This parameter disables movement based on the G28 command to a reference position. So, use this parameter only in special cases.

#1 RF2x If G28 is specified for an axis for which a reference position is already established (ZRF = 1) when a linear scale with an absolute address zero point or a linear scale with absolute address reference marks is used:
0: A movement is made to the reference position.
1: No movement is made to the intermediate position and reference position, but the operation is completed.

> **NOTE**
> This parameter disables movement based on the G28 command to a reference position. So, use this parameter only in special cases.

	#7	#6	#5	#4	#3	#2	#1	#0
1819						DATx		

[Input type] Parameter input
[Data type] Bit axis

#2 DATx When a linear scale with an absolute address zero point or a linear scale with absolute address reference marks is used, the automatic setting of parameters Nos. 1883 and 1884 at manual reference position return time is:
0: Not performed.
1: Performed.
The automatic setting procedure is as follows:
<1> Set an appropriate value in parameters Nos. 1815, 1821, and 1882.
<2> Position the machine at the reference position by manual operation.
<3> Set this parameter to 1.
Alarm PS5220, "REFERENCE POINT ADJUSTMENT MODE", occurs.
<4> Perform a manual reference position return operation. Upon completion of manual reference position return operation, parameters Nos. 1883 and 1884 are set, and this parameter is automatically set to 0.
Make a reset to release alarm PS5220.

1869	The amount of one rotation of rotary axis B type (each axis)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (Refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Normally, the amount of one rotation of rotary axis is 360, and the machine position 0 is the reference position.
In this case, this parameter is set to 0.

For instance, when this parameter is set to 523.000, the amount of one rotation become 523.000 (in the case of IS-B), if it is necessary to set it arbitrarily.

NOTE
1 This parameter is available for only the rotary axis B type with an absolute position detector (absolute Pulsecoder) or a rotary scale with distance-coded reference marks (serial), as for the bit 3 (SCRx) of parameter No. 1817 is set to 1 or the bit 0 (RVSx) of parameter No. 1815 is set to 1.
2 In the case that the amount of one rotation of rotary axis is 360, this parameter is set to 0.If it is necessary to set an amount of one rotation of rotary axis arbitrarily, this parameter is set to the amount of one rotation.

> **NOTE**
> 3 When this parameter is set, machine position and position on absolute position detector become uncorresponding. So, establish the reference position again. (Consequently, the bit 4 (APZx) of parameter No. 1815 (indicating that the correspondence is established) is set to 0, and an alarm DS0300, “APC ALARM: NEED REF RETURN”, occurs. Why the bit 4 (APZx) of parameter No. 1815 is set to 0 can be checked using bit 0 of diagnosis data No.310).
> 4 This parameter No. 1869 is common in movable range that is under one rotation (the bit 3 (SCRx) of parameter No. 1817 is set to 1) and movable range that is over one rotation (the bit 0 (RVSx) of parameter No. 1815 is set to 1).
> 5 Please set 0 to this parameter if the distance coded rotary scale interface (phase A/B) is used.

1821	Reference counter size for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

Set the interval of reference mark 1 of distance coded linear scale.

1882	Interval of reference mark 2

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

Set the interval of reference mark 2 of distance coded linear scale.

1883	Distance 1 from the scale zero point to reference position

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999999999 to 999999999

1884	Distance 2 from the scale zero point to reference position

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

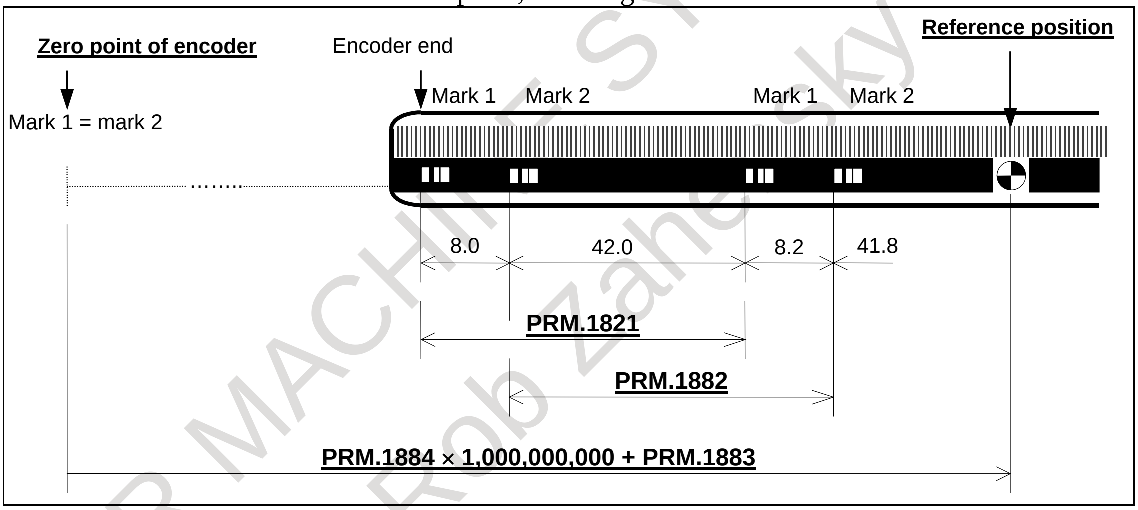
[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999 to 999

When a linear scale with absolute address reference marks is used, set the distance from the scale zero point to reference position in parameter Nos. 1883 and 1884).

Distance from the zero point to the reference position of a linear scale
= No. 1884 × 1,000,000,000 + No. 1883

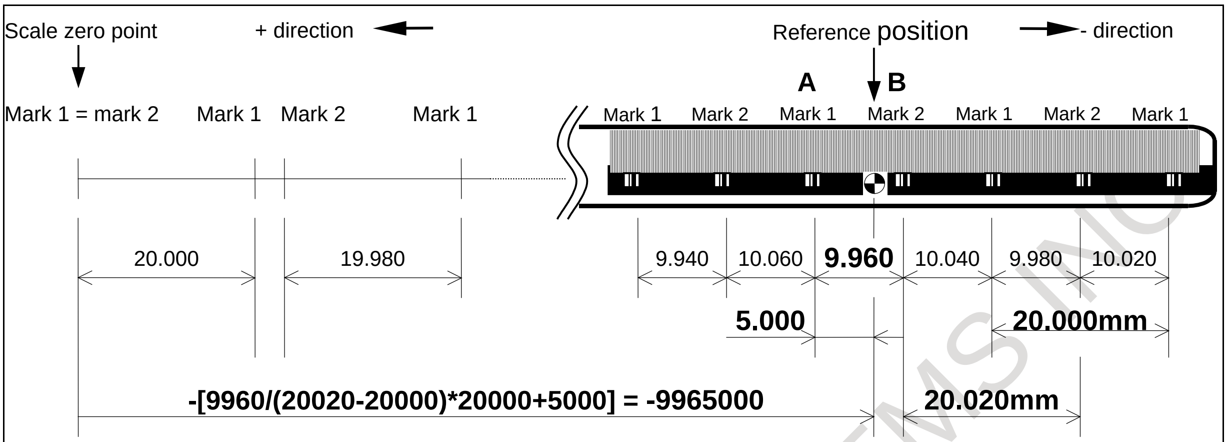
The scale zero point represents a point where mark 1 and mark 2 match. Usually, this point is a virtual point that does not physically exist on the scale. (See the following figure.)

If the reference position is placed in the + direction when viewed from the scale zero point, set a positive value. If the reference position is placed in the - direction when viewed from the scale zero point, set a negative value.



[Example of parameter settings]

When an encoder as shown the following figure is used with an IS-B, millimeter machine:



Parameters

No.1821 (interval of mark 1) = 20000
No.1882 (interval of mark 2) = 20020
No.1883 (reference position) = position of point A + 5.000
= distance between A and B/(mark 2 – mark 1) × mark 1 + 5000
= 9960/(20020-20000) × 20000 + 5000
= 9965000
= -9965000 (the reference position is on the negative side)

[Setting parameter No. 1883]

When it is difficult to measure the distance from the scale zero point to the reference position (parameter No. 1883), the method described below can be used to find the distance.

<1> Set parameter No. 1815 to enable this function.
Set an appropriate value in parameter No. 1821 and No. 1882.
Parameter No. 1240 is set to 0.
Parameters Nos. 1883 and 1884 are set to 0.
<2> At an appropriate position, establish a reference position.
(As a result, the machine coordinate represents the distance from the scale zero point to the current position.)
<3> By jog feed or handle feed, place the machine at the accurate reference position.
<4> In parameter No. 1883, set the machine coordinate of that time converted to the detection unit (machine coordinate × CMR).
<5> If necessary, set parameter No. 1240.

NOTE

1 Set parameter Nos. 1883 and 1884 so that the distance from the scale zero point to the reference position is within the range from -999,999,999,999 to +999,999,999,999. If a value beyond this range is set, an alarm DS1448, "ILLEGAL PARAMETER (D.C.S.)", is issued.
2 The scale area on the scale cannot be extended across the scale zero point. Make parameter settings not to cause the scale area to extend beyond the scale zero point.

Alarm and message

Number	Message	Description
PS5220	REFERENCE POINT ADJUSTMENT MODE	In case of distance coded linear scale I/F, the reference point auto setting bit 2 (DATx) of parameter No. 1819 is set to 1. Move the machine to reference position by manual operation and execute manual reference return.
SV1051	ILLEGAL SYNCHRONOUS AXIS	In axis synchronous control, the parameter setting for the axis for which distance coded linear scale I/F is used, is incorrect. Set the following parameters to the same values for the master axis and slave axis. - Parameter No. 1821 (mark-1 interval) - Parameter No. 1882 (mark-2 interval) - Bit 2 (DC2x) and Bit 1 (DC4x) of parameter No.1802 (the number of marks)
DS0020	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the perpendicular axis before the completion of a return to the reference position on the angular axis. However, this attempt failed because a manual return to the reference position during angular axis control or an automatic return to the reference position after power-up was not commanded. First, return to the reference position on the angular axis, then return to the reference position on the perpendicular axis.
DS0026	MISMATCH OF ANGULAR AXIS(D.C.S)	On angular axis control, one of the angular/perpendicular axes is the scale with ref-pos, and the other of them is not the scale with ref-pos. Such system is not admired.
DS0027	MISMATCH OF SYNCHRONOUS AXIS(D.C.S)	- Master/slave axes of axis synchronous control, one of them is the linear scale with distance-coded reference marks, and the other of them is not the linear scale with distance-coded reference marks. Please establish reference position with the input signal SYNCn<Gn138>, SYNCJn<Gn140> or parameter setting to 0. - The slave axis of the axis synchronous control is two or more settings. Such system is not admired.
DS1448	ILLEGAL PARAMETER (D.C.S.)	The setting value of parameter for distance coded linear scale I/F is satisfied the following any conditions. - The absolute-position detection function is enabled. - Either parameter No. 1821 (mark-1 interval) or parameter 1882 (mark-2 interval) is set to 0. - Parameters Nos. 1821 and 1882 have identical settings. - The difference between the settings made for parameters Nos. 1821 and 1882 is greater than or equal to twice either setting. - The setting value of parameters 1883 and 1884 are over the valid data range. - In case of Rotary axis (B type), the values other than 0 are set to the parameter No.1868 and No.1869 or bit 0 (RVSx) of parameter No.1815 is set to 1.
DS1449	REFERENCE MARK ARE DIFFERENT FROM PARAMETER	In case of distance coded linear scale I/F, the actual interval of reference marks is different from setting value in parameters Nos. 1821 and 1882.

Diagnosis data

3545	Measurement point 1 of the reference marks of distance coded linear scale
3546	Measurement point 2 of the reference marks of distance coded linear scale
3547	Measurement point 3 of the reference marks of distance coded linear scale
3548	Measurement point 4 of the reference marks of distance coded linear scale

[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999999999 to 999999999

3549	Internal status of the reference marks of distance coded linear scale
3550	Scale counter value of the reference marks of distance coded linear scale

[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999999999 to 999999999

3551	Scale counter value of the reference marks of distance coded linear scale (high)

[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999 to 999

Scale counter value of the reference marks of distance coded linear scale = diagnosis data No.3551 × 1,000,000,000 + diagnosis data No.3550

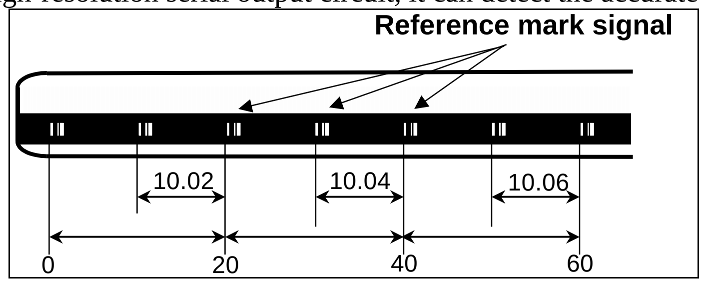
4.9 LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL)

Overview

By using High-resolution serial output circuit for the linear scale with distance-coded reference marks (serial), the CNC measures the interval of referenced mark by axis moving of short distance and determines the absolute position.
This function enables high-speed high-precision detection by using High-resolution serial output circuit.
It is available that using maximum stroke 30 meters length.

Explanation

The linear scale with distance-coded reference marks (serial) is combined the irregular reference marked linear scale with the High-resolution serial output circuit, it can detect the accurate position.



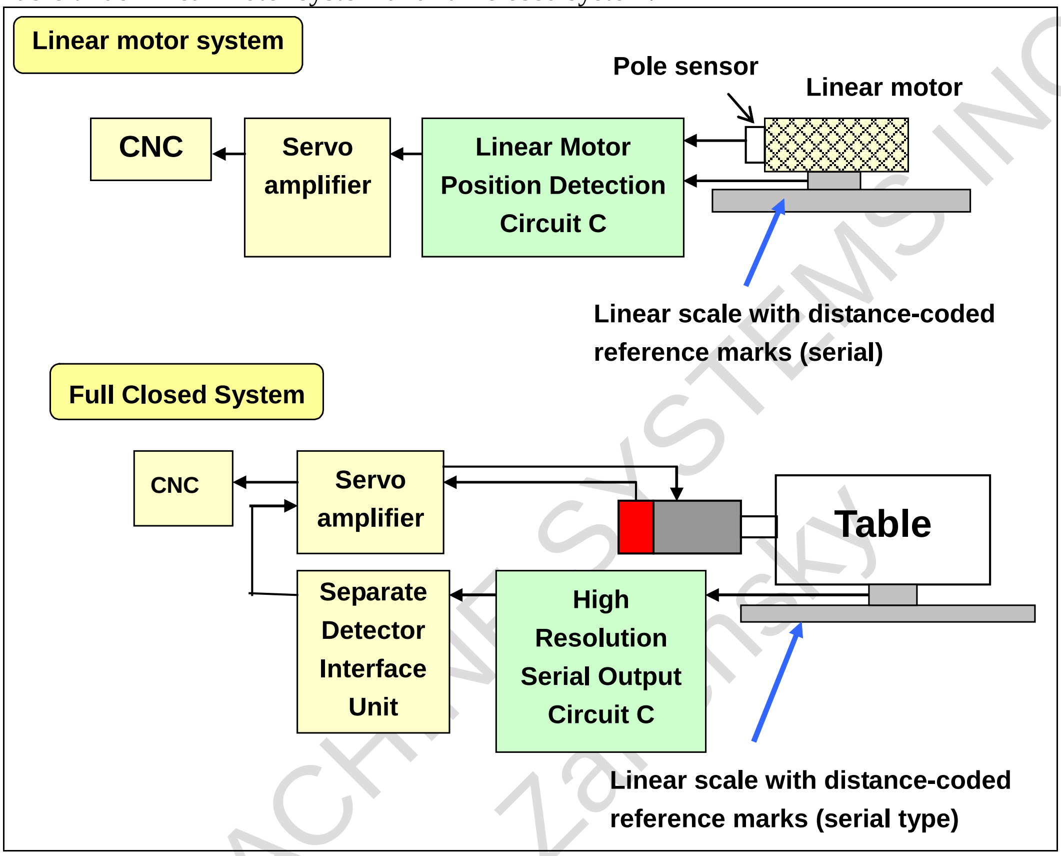
The CNC measures the interval of referenced mark by axis moving of short distance and determines the absolute position, because of the interval of each reference mark is different with regular interval.

It is not necessary that the axis is moved to the reference position for establishment of reference position.

This function enables high-speed high-precision detection by using High-resolution serial output circuit. It is available that using maximum stroke 30 meters length.

- Connection

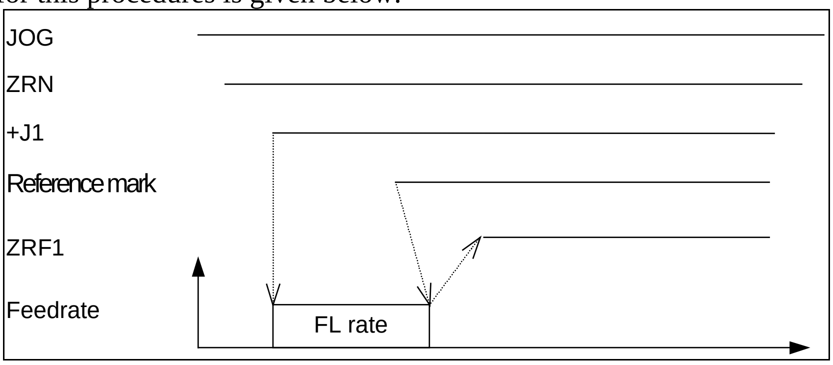
It is available under linear motor system and full closed system.



- Procedure for reference position establishment through manual operation

(1) Select the JOG mode, and set the manual reference position return selection signal ZRN to "1".
(2) Set the feed axis and direction selection signal (+J1, -J1, +J2, -J2, …) of a target axis to "1" and feed toward the reference position.
(3) The axis is fed at a constant low speed (reference position return FL feedrate specified by parameter No.1425 setting).
(4) When the absolute position of linear scale with distance-coded reference marks (serial) is detected, the axis stops. Then the absolute position of CNC is calculated and the reference position establishment signal (ZRF1, ZRF2, ZRF3, …) turns to "1".

The timing chart for this procedures is given below.



- Procedure for reference position establishment through automatic operation

If an automatic reference position return (G28) is specified before a reference position is not established, steps (3) to (4) above are performed automatically.
After the reference position is established, the automatic reference position return is performed by setting of bit 0 (RFS) of parameter No. 1818.

- Stopping the operation for establishing a reference position

The operation for establishing a reference position is stopped if any of the following operations is performed in steps (3) to (4), described above.

- Reset
- Setting the feed axis and direction selection signal (+J1, -J1, +J2, -J2, etc.) to "0"
- Setting the Servo off signals (SVF1, SVF2, etc.) to "1"

If any of the following operations is performed during the operation of automatic reference position return (G28) before a reference position is not established, the operation for establishing a reference position stops:

- Reset
- Performing feed hold during movement from an intermediate position to the reference position
- Setting the Servo off signals (SVF1, SVF2, etc.) to "1"

If the operation for establishing a reference position is stopped by an operation other than a reset, the operation for establishing a reference position must be reset and resumed.

- Establishing a reference position and moving to the reference position

By following operation, establishing a reference position and moving to the reference position is performed.

	Moving through manual operation in REF mode	Moving through automatic operation by automatic reference position return (G28)
The reference position is not established.	Establishing the reference position	Firstly, moving to the intermediate position, and establishing the reference position. Secondly, whether moving to the reference position or not is performed by setting bit 0 (RFS) of parameter No.1818.
The reference position is established.	Moving to the reference position	Whether moving to the intermediate position and the reference position or not is performed by setting bit 1 (RF2) of parameter No.1818.

- Axis synchronous control

In case of using the axis synchronous control, please confirm the following items.

- When this function is used with axis synchronous control axes, the linear scale with distance-coded reference marks (serial) used for the master axis and that used for the slave axis must have reference marks placed at identical intervals.
- The linear scale with distance-coded reference marks (serial) can be installed on the master and slave axes either in the same or opposite direction.
- To the axis parameters, which relate to this function (except No.1883, No.1884) (distance from the scale zero point to reference position), the same value must be set for the master axis and for the slave axis.
- The linear scale with distance-coded reference marks (serial) should be applied for the master axis and the slave axis.
  If either of the master axis or the slave axis is not the linear scale with distance-coded reference marks (serial), alarm DS0018, "SERIAL DCL:MISMATCH (SSYNC CTRL)", occurs when reference position establishment is tried by the axis synchronous control.
- During operating the establishment of reference position, the state of synchronous control axis selection signals (SYNCn<Gn138> or SYNCJn<Gn140>) should be kept.

Both of synchronous axes (master axis and slave axis) are fed on the reference position return FL feedrate until distance coded scales of both axes detect the absolute position.
Then absolute position of both axes are calculated and the reference position establishment signal (ZRF1, ZRF2, ...) turn to "1".

> **NOTE**
> When this function is used by the axis synchronous control, only one axis is available for the slave axis.

- Angular axis control

In case of using the angular axis control, please confirm the following items.

- It is necessary to use the linear scale with distance-coded reference marks (serial) for both the perpendicular axis and the angular axis.
  If not, the alarm DS0019, "2SERIAL DCL:MISMATCH(ANGL-AXIS)", occurs when reference position establishment is commanded.
- Please set so that whenever the reference position establishment of angular axes is tried, perpendicular axes are fed (bit 2 (AZR) of parameter No. 8200 to 0 and input signal NOZAGC <G063.5> to "0").
  If not, the alarm DS0019 occurs when reference position establishment is commanded.
- When the reference point of the perpendicular axis is established, it is necessary to establish the reference point of the angular axis previously. When the perpendicular axis is commanded previously or the perpendicular axis and the angular axis are commanded simultaneously, the alarm DS0020, "REFERENCE RETURN INCOMPLETE", occurs.
- On angular axis control, if you use automatic setting of parameter Nos.1883 and 1884 on reference position establishment (bit 2 (DATx) of parameter No.1819=1) for both the perpendicular axis and the angular axis, please establish reference point of perpendicular axis after reference position auto establishment and return of angular axis.

In manual reference position return, the perpendicular axis cannot be specified while the angular axis reference point is being established. The perpendicular axis, if specified, is ignored.

Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
1815						DCLx	OPTx	

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#1 OPTx The separate position detector is:
0: Not to be used (semi-closed system)
1: To be used (full-closed system)

> **NOTE**
> Set this parameter to 1 when using a linear scale with distance-coded reference marks (serial) (full-closed system).

#2 DCLx As a separate position detector, a linear scale with reference marks or a linear scale with an absolute address zero point is:
0: Not used.

1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
1818					SDCx		RF2x	RFSx

[Input type] Parameter input
[Data type] Bit axis

#0 RFSx If G28 is specified for an axis for which a reference position is not established (ZRF = 0) when a linear scale with an absolute address zero point is used:
0: A movement is made to the reference position after reference position establishment operation.
1: No movement is made after reference position establishment operation, but the operation is completed.

> **NOTE**
> This parameter disables movement based on the G28 command to a reference position. So, use this parameter only in special cases.

#1 RF2x If G28 is specified for an axis for which a reference position is already established (ZRF = 1) when a linear scale with an absolute address zero point is used:
0: A movement is made to the reference position.
1: No movement is made to the intermediate position and reference position, but the operation is completed.

> **NOTE**
> This parameter disables movement based on the G28 command to a reference position. So, use this parameter only in special cases.

#3 SDCx A linear scale with an absolute address zero point is:
0: Not used.
1: Used.

> **NOTE**
> 1 When the parameter SDCx is set, the power must be turned off before operation is continued. Alarm PW0000, “POWER MUST BE OFF ” does not occur.
> 2 Please set bit 1 (SDC) of parameter No. 1815 to 1 when the full closed system. But please set this parameter to 0 when the Linear motor system.

	#7	#6	#5	#4	#3	#2	#1	#0
1819						DATx		

[Input type] Parameter input
[Data type] Bit axis

#2 DATx The automatic setting of parameters Nos. 1883 and 1884 at manual reference position return time is:
0: Not performed.
1: Performed.

The automatic setting procedure is as follows:
<1> Set an appropriate value in parameters Nos. 1815, 1821, and 1882.
(In case of the linear scale with distance-coded reference marks (serial), it is not necessary to set the parameter No.1882.)
<2> Position the machine at the reference position by manual operation.
<3> Set this parameter to 1.
Alarm PS5220, "REFERENCE POINT ADJUSTMENT MODE", occurs.
<4> Perform a manual reference position return operation. Upon completion of manual reference position return operation, parameters Nos. 1883 and 1884 are set, and this parameter is automatically set to 0.
Make a reset to release alarm PS5220.

1883	Distance 1 from the base point to reference position

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999999999 to 999999999

1884	Distance 2 from the base point to reference position

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999 to 999
Set the distance from the base point to the reference position in parameter Nos. 1883 and 1884. The base point is a point at a scale end as shown Fig. 4.9 (a).

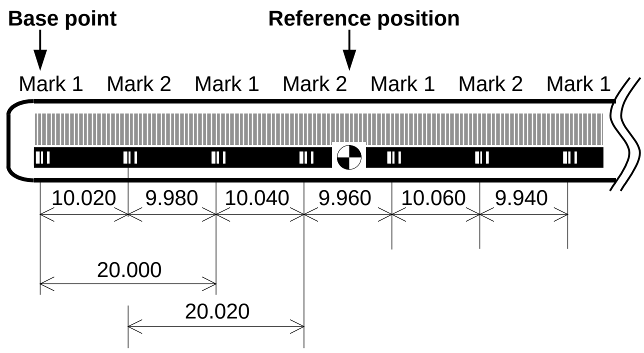


Fig. 4.9 (a)

If the reference position is located in the positive direction when viewed from the base point, set a positive value; if the reference position is located in the negative direction, set a negative value. Set the value by following the steps explained below.

<1> Set bit 1 (OPT) of parameter No. 1815 , bit 2 (DCL) of parameter No. 1815, and bit 3 (SDC) of parameter No. 1818 to enable this function.
Set 0 in parameter No. 1240.
Set 0 in parameter Nos. 1883 and 1884.
<2> At an appropriate position, establish a reference position.
(Consequently, the machine coordinate value indicates the distance from the base point to current position.)
<3> By jog feed or handle feed, place the machine at the accurate reference position.
<4> In parameters Nos. 1883 and 1884, set the machine coordinate of that time converted to the detection unit (machine coordinate × CMR).

If necessary, set parameter No. 1240.

NOTE
1. Set parameter Nos. 1883 and 1884 so that the distance from the base point to the reference position is within the range from -999,999,999,999 to +999,999,999,999. If a value beyond this range is set, an alarm DS0016, "SERIAL DCL:FOLLOW-UP ERROR", is issued.
2. The scale area on the scale cannot be extended across the base point. Make parameter settings not to cause the scale area to extend beyond the base point.

14010	Maximum allowable travel distance when the reference position is established for a linear scale with an absolute address reference position

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

This parameter sets the maximum allowable travel distance at the FL rate when the reference position is established for a linear scale with an absolute address reference position. When the travel distance exceeds the setting of this parameter, the alarm DS0017, “SERIAL DCL:REF-POS ESTABLISH ERR” is issued. When this parameter is set to 0, the maximum allowable travel distance is not checked.

NOTE
1. To establish the reference position with axis synchronous control, set the parameter for both master and slave axes.
2. In angular axis control, the setting of this parameter is invalid to the orthogonal axis where the reference position on the angular axis is being established.

Alarm and message

No.	Message	Description
PS0372	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the orthogonal axis before the completion of a return to the reference position on the angular axis. However, this attempt failed because a manual return to the reference position during angular axis control or an automatic return to the reference position after power-up was not commanded. First, return to the reference position on the angular axis, then return to the reference position on the orthogonal axis.

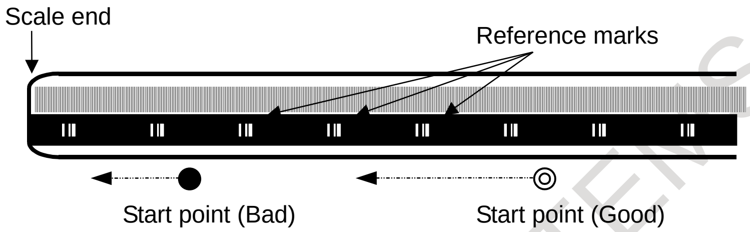
No.	Message	Description
PS0376	SERIAL DCL: ILLEGAL PARAMETER	1. When bit 1 (OPTx) of parameter No. 1815 is set to 1, bit 3 (PFSE) of parameter No. 2002 is set to 0 2. The absolute-position detection function is enabled. (Bit 5 (APCx) of parameter No. 1815 is set to 1.)
PS5220	REFERENCE POINT ADJUSTMENT MODE	In case of distance coded linear scale I/F, the reference point auto setting bit 2 (DATx) of parameter No. 1819 is set to 1. Move the machine to reference position by manual operation and execute manual reference return.
DS0016	SERIAL DCL:FOLLOW-UP ERROR	1. The specification of parameter No. 1883 or 1884 is out of range. 2. During the establishment of an origin, the distance between the current position and the reference position (detection unit) exceeded ±2147483647. To avoid this situation, modify either the current position or the reference position.
DS0017	SERIAL DCL:REF-POS ESTABLISH ERR	During the establishment of an origin, the amount of travel at the FL feedrate exceeded the setting of parameter No. 14010.
DS0018	SERIAL DCL:MISMATCH (SSYNC CTRL)	Master/slave axes of axis synchronous control, one of them is the linear scale with distance-coded reference marks (serial) and the other of them is not the linear scale with distance-coded reference marks (serial). Please establish reference position with the input signal SYNCn<G138>, SYNCJn<G140> or bit 5 (SCA) of parameter No. 8304 setting to 0.
DS0019	SERIAL DCL:MISMATCH(ANGL-AXIS)	On angular axis control, one of the angular/perpendicular axes is the scale with ref-pos, and the other of them is not the scale with ref-pos. Such system is not admired.
DS0020	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the orthogonal axis before the completion of a return to the reference position on the angular axis. However, this attempt failed because a manual return to the reference position during angular axis control or an automatic return to the reference position after power-up was not commanded. First, return to the reference position on the angular axis, then return to the reference position on the orthogonal axis.
SV0366	PULSE MISS(INT)	A pulse error occurred on the built-in Pulsecoder.
SV0367	COUNT MISS(INT)	A count error occurred on the built-in Pulsecoder.
SV0382	COUNT MISS(EXT)	A count error occurred on the separate detector.
SV0383	PULSE MISS(EXT)	A pulse error occurred on the separate detector.

Caution

> ⚠ **CAUTION**
> 1 When the Linear scale with distance-coded reference marks (serial) is used, set bit 3 (SDCx) of parameter No. 1818 of the corresponding axis to 1.

⚠ CAUTION

2. On the Linear scale with distance-coded reference marks (serial), the axis does not stop until three reference marks are detected. If this procedure is started at the position near the scale end, CNC can not detect three reference marks and the axis does not stop until over travel alarm occurs. Please care to start at the position that has enough distance from scale end.
   And if establishment of reference position is failed, the establishment is retried. Then axis does not stop until still more three reference marks are detected. So set the maximum move amount (detection unit: parameter No.14010) not to reach the scale end.



3. On flexible synchronization mode, reference position can't be established.
4. In case that straightness compensation function is used, the movement is explained below.
   When the reference position establishment of moving axis is executed after the establishment of compensation axis after power on, the compensation axis is moved by straightness compensation amount when the reference point of moving axis is established.
5. It is not available to use this function and the temporary absolute coordinate setting together.
6. This function cannot be used together with synchronous control, composite control, and superimposed control.
7. When the axis used this function, the following function can not be used.
   - Absolute position detection

4.10 EXTENDED FUNCTION OF THE DISTANCE CODED LINEAR SCALE INTERFACE

Overview

The distance coded linear scale interface has reference marks at intervals that change at a constant rate. By determining the reference mark interval, the corresponding absolute position can be deduced. When a G00 command or a move command in jog feed is specified for an axis for which the distance coded linear scale interface is used, this function establishes the reference position by measuring the reference mark intervals automatically. Therefore, after CNC power-up, the reference position can be established without performing reference position return operation.

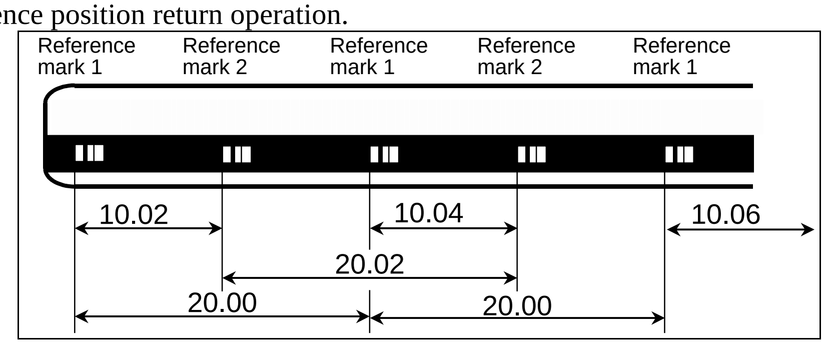


Fig. 4.10 (a) Sample distance coded linear scale interface

To use this function, the distance coded linear scale interface is also required.

4.10.1 Reference Position Established by the G00 Command

Explanation

- Activation conditions

When the following conditions are satisfied, reference position establishment operation is performed automatically:

<1> A G00 command is specified to cause a movement along an axis for which no reference position has been established.

<2> The movement direction in <1> above matches the reference position return direction set by bit 5 (ZMI) of parameter No. 1006.

<3> The specified axis is not in the following modes:

- 3-dimensional coordinate conversion
- Coordinate system rotation
- Mirror image (mirror image by signal or setting)
- Programmable mirror image
- Scaling (M series)
- AI contour control mode
- Mirror image of facing tool posts

> **NOTE**
> If all the above conditions are not satisfied, the reference position establishment operation is not performed, and normal G00 command operation is performed.

- Operation

The reference position establishment procedure is explained below.

<1> The tool is fed along a specified axis at the reference position return FL feedrate (parameter No. 1425).

<2> Upon detection of a reference mark on the scale, the tool is stopped briefly then fed at the reference position return FL feedrate.

<3> Step <2> above is repeated until two, three, or four reference marks are detected on the scale. Then, the reference position is established, and the reference position establishment signal (ZRF1, ZRF2, ZRF3, etc.) is set to "1".

<4> The tool is fed to a specified end point at a rapid traverse rate.

A time chart for the above procedure is shown below.

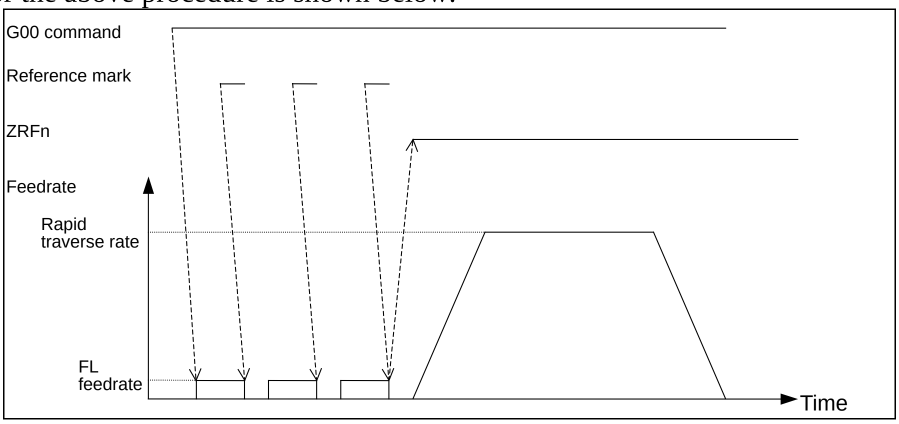


Fig. 4.10.1 (a) Time chart for reference position establishment (G00)

The specifications for the steps of detecting reference marks and establishing the reference position (steps <1> to <3> above) are the same as for the conventional distance coded linear scale interface. The restrictions are also the same.
For details, refer to " Linear scale with distance-coded reference marks (serial)."

- Tool path

The tool path in the G00 command is explained below.

(1) When no axis requires the reference position establishment When the reference position has already been established for all specified axes, the reference position establishment operation is not performed.
For example, suppose that the reference position is already established for the X-, Y-, and Z-axes, and that G00 Xxx Yyy Zzz; is specified. Then, normal rapid traverse operation takes place. The tool path follows the setting in bit 1 (LRP) of parameter No. 1401.

(2) When all axes require the reference position establishment operation
Suppose that the reference position is not established for the X-, Y-, and Z-axes and that G00 Xxx Yyy Zzz; is specified. The operation in this case is shown in the Fig. 4.10.1 (b).
Operation 1 in the figure establishes the reference position. During the establishment operation, the tool path is always of the non-linear interpolation type regardless of the setting in bit 1 (LRP) of parameter No. 1401.
Operation 2 performs positioning to a specified end point. During this operation, the tool path follows the setting in bit 1 (LRP) of parameter No. 1401.
After operation 1 is completed for all axes, operation 2 starts.

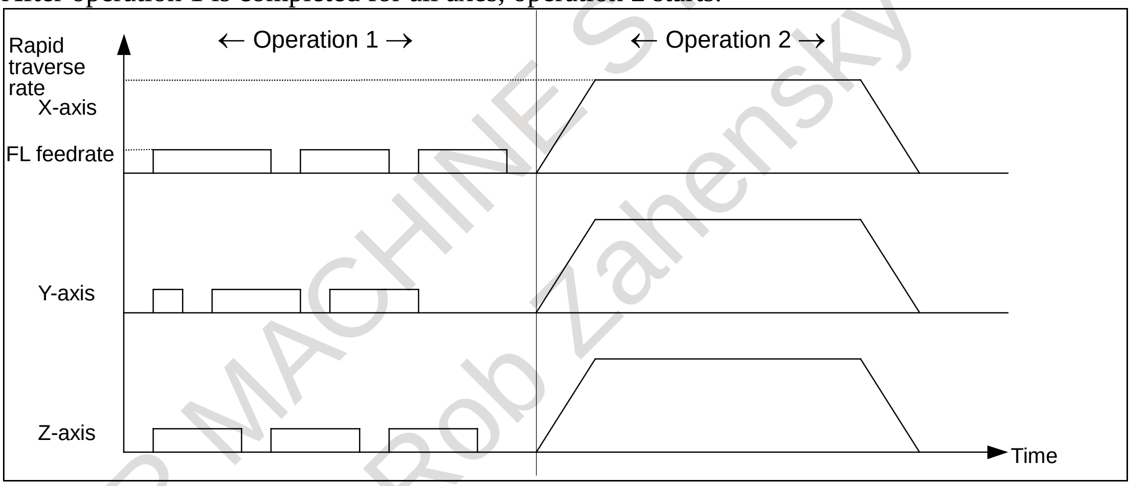


Fig. 4.10.1 (b) When the reference position is established for all axes

(3) When some axes require the reference position establishment operation and others do not require the establishment operation
For example, suppose that the reference position is already established for the X-axis and that the reference position is not yet established for the Y- and Z-axes. Also suppose that G00 Xxx Yyy Zzz; is specified. The operation in this case is shown in the Fig. 4.10.1 (c).
In operation 1 in the figure, movement to a specified position is made along the X-axis for which the reference position is already established. For the Y- and Z-axes for which no reference position is established, the reference position is established. During the establishment operation, the tool path is always of the non-linear interpolation type regardless of the setting in bit 1 (LRP) of parameter No. 1401.
In operation 2, positioning to a specified end point is performed along the Y- and Z-axes. The tool path along the Y- and Z-axes then follows the setting in bit 1 (LRP) of parameter No. 1401. Because positioning to the specified position is already made along the X-axis, no movement is made along the X-axis.
After operation 1 is completed for all axes, operation 2 starts.

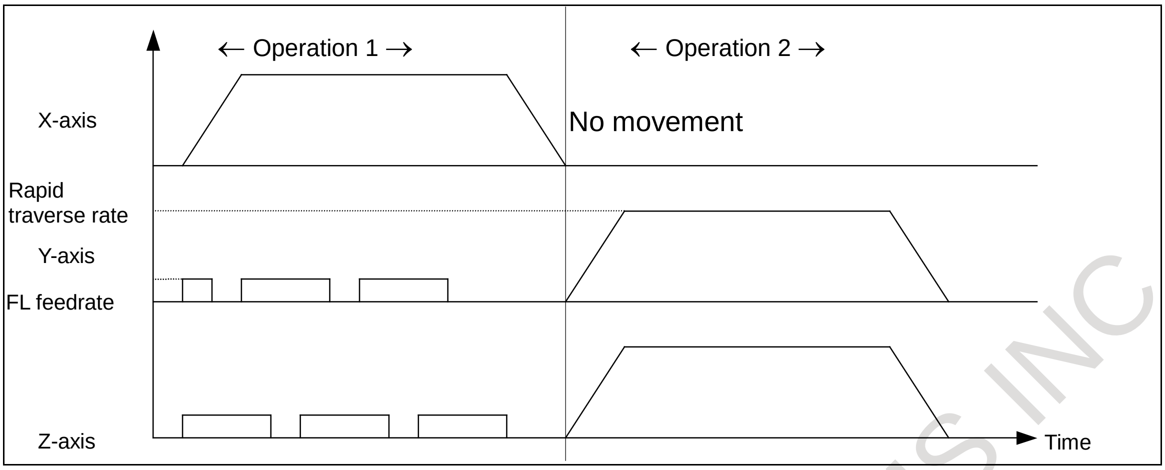


Fig. 4.10.1 (c) When an axis does not require the reference position establishment operation and others require the establishment operation

- Absolute command and incremental command

After the reference position has been established, positioning to a specified end point is performed. This operation is explained below.

(1) When an absolute command is specified
Movement to the end point in the new coordinate system that has been established in step <3> is made. For example, when G90 G00 Xxx.Yyy.; is specified, the tool moves as indicated with the bold line in the Fig. 4.10.1 (d). Note that, however, the figure shows the positional relationship among points and that the intermediate tool path is not always of the linear interpolation type.

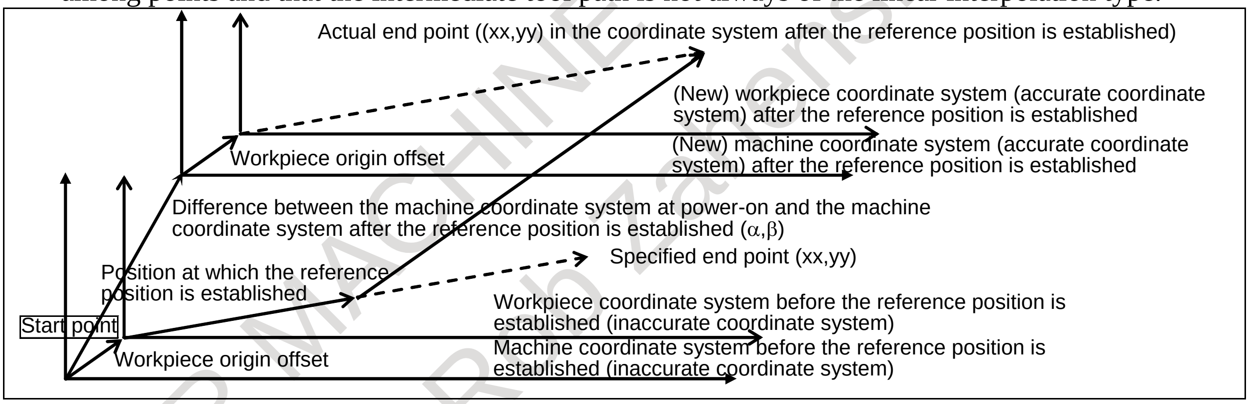


Fig. 4.10.1 (d) Operation when an absolute command is specified

(2) When an incremental command is specified
Movement is made along each axis by a specified distance. (The movement is indicated with the bold line in the Fig. 4.10.1 (e). Note that the intermediate tool path is not always of the linear interpolation type.) The coordinates at the end point are those in a newly established coordinate system. When there is a difference (α, β) between the newly established coordinate system and the old coordinate system, the coordinates at the end point are shifted by (α, β). For example, when G91 G00 X100.0Y30.0; is specified from the position of which absolute coordinates are (0,0), the distance of movement along each axis is (100,30), and the coordinates of the end point are (100-α, 30-β).

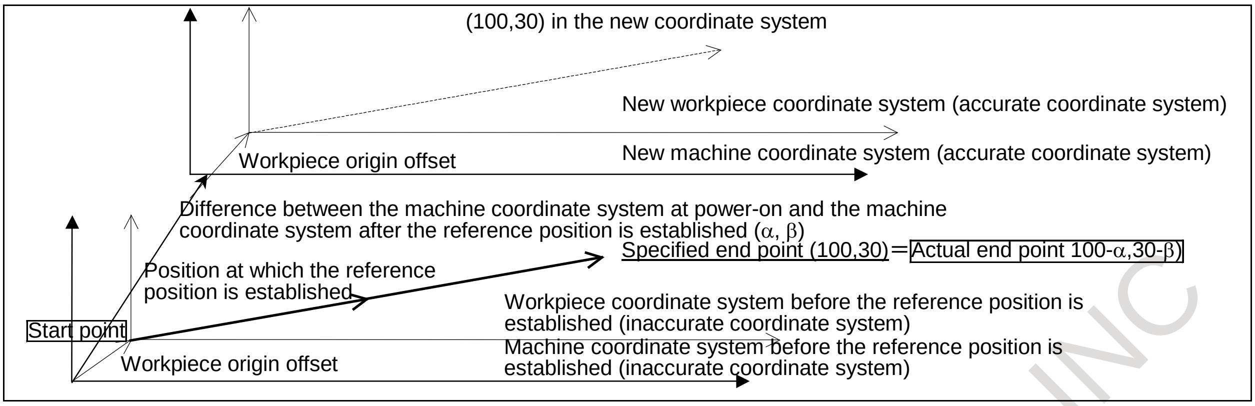


Fig. 4.10.1 (e) Operation when an incremental command is specified

- When a short distance is specified

When a short distance is specified, the end point can be reached before two, three, or four reference marks are passed. Even in such a case, the CNC makes a movement at the FL feedrate while detecting reference marks, but the CNC does not establish the reference position even when the end point is reached.
If the reference position is not established, the next G00 command causes the reference position establishment operation again. The CNC does not use data on the reference marks detected through the previous movement and detects three or four reference marks again to establish the reference position.

- Interruption by feed hold

When a feed hold has been applied during reference position establishment operation, the reference position establishment operation is not performed after execution is restarted. After the restart, non-linear interpolation type positioning is performed. In this block, the reference position is not established, so reference position establishment operation is performed again when the next G00 command is specified.

- Interruption by reset or emergency stop

When a reset or emergency stop is applied during reference position establishment operation, the reference position establishment operation is interrupted. Since the reference position is not established, reference position establishment operation is performed again when the next G00 command is specified.

- When an illegal reference mark interval is detected

If a correct reference mark interval cannot be detected for a cause, the tool is positioned to the end point without establishing the reference position. Therefore, the machine position, absolute coordinates, and machine coordinates of the end point are set as if a G00 command without reference position establishment operation were executed. However, the intermediate tool path is different from the tool path obtained by executing the G00 command.
Since the reference position is not established, reference position establishment operation is performed again when the next G00 command is specified.
Even when the CNC detects an illegal reference mark interval, it does not issue the alarm DS1449 "REFERENCE MARK ARE DIFFERENT FROM PARAMETER".

4.10.2 Reference Position Establishment by Jog Feed

Explanation

- Activation condition

If the following conditions are satisfied, reference position establishment operation is performed automatically:

<1> For an axis for which no reference position has been established, feed axis and direction selection signal +Jn or -Jn <G100, G102> is input in jog mode.

<2> The move direction in <1> above matches the reference position return direction set by bit 5 (ZMI) of parameter No. 1006.

<3> The specified axis is not in the following modes:
- 3-dimensional coordinate conversion
- Mirror image (mirror image by signal or setting)

> **NOTE**
> If all the above conditions are not satisfied, reference position establishment operation is not performed, and the same operation as normal jog feed is performed.

- Operation

The reference position establishment procedure is explained below.

<1> A movement on the specified axis starts at the FL feedrate of reference position return (parameter No.1425).

<2> Upon detection of a reference mark on the scale, the tool is stopped briefly then fed at the reference position return FL feedrate.

<3> Step <2> above is repeated until two, three, or four reference marks are detected on the scale. Then, the reference position is established, and the reference position established establishment signal (ZRF1, ZRF2, ZRF3, etc.) is set to "1".

<4> The tool is fed in the direction selected by the feed axis direction selection signal at the jog feedrate.

When the feed axis and direction selection signal is set to "0" during steps <2> to <4>, feed operation stops. When the feed axis direction selection signal is set to "1" again, the reference position is established. A time chart for the above procedure is shown below.

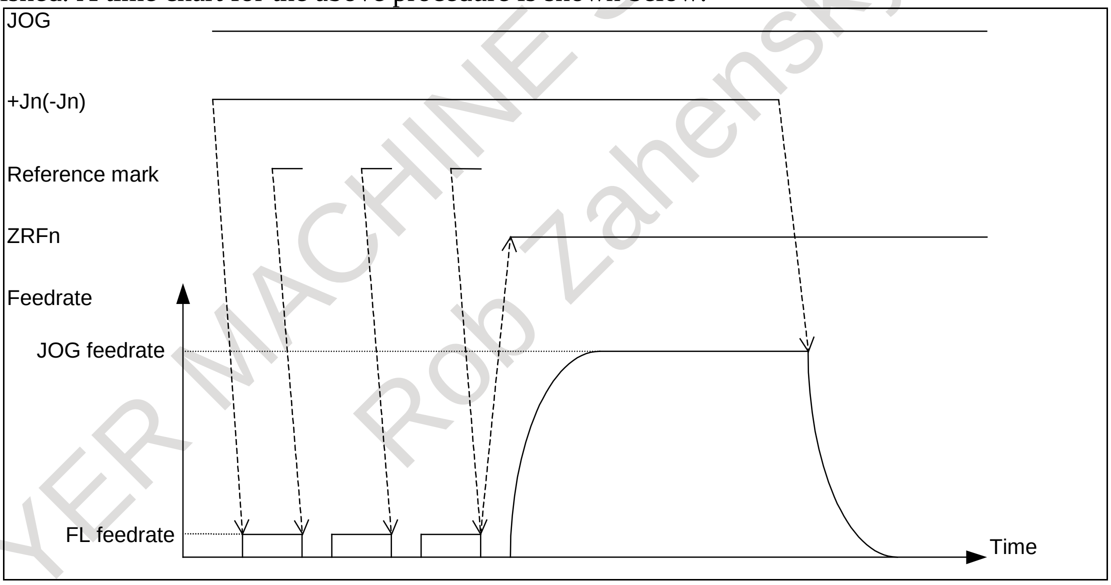


Fig. 4.10.2 (a) Time chart for reference position establishment (jog feed)

- Interruption by the feed axis and direction selection signal

If the feed axis and direction selection signal +Jn (-Jn) is set to "0" during reference position establishment, the reference position establishment operation is interrupted. In this case, data on the reference marks detected through the previous movement is not used for the next reference establishment operation. When the feed axis and direction selection signal +Jn (-Jn) is set to "1" again, the reference position establishment operation is resumed, and the reference position is established after two, three, or four reference marks are detected.

- Interruption by reset or emergency stop

When a reset or emergency stop is applied during reference position establishment operation, axis movement stops, and the reference position establishment operation is interrupted. In this case, data on the reference marks detected through the previous movement is not used for the next reference establishment operation. When the feed axis and direction selection signal +Jn (-Jn) is set to "1" again, the reference position is established after two, three, or four reference marks are detected.

Caution

- PMC axis control

The reference position establishment operation is not performed in rapid traverse (axis control command 00h) or continuous feed (axis control command 06h) under PMC axis control.

- Rapid traverse operation other than G00

In a rapid traverse operation automatically generated by a command such as canned cycle, no reference position establishment operation is carried out.

- Reference position establishment operation

This function has the same specifications as the conventional function of the distance coded linear scale interface, with regard to the operation to detect a reference mark and establish a reference position. The parameter setting, limitation, and the like are also the same.
For details, refer to " Linear scale with distance-coded reference marks (serial)."

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1802						DC2x	DC4x	

[Input type] Parameter input
[Data type] Bit axis

#1 DC4x When the reference position is established on the linear scale with reference marks:
0: An absolute position is established by detecting three reference marks.
1: An absolute position is established by detecting four reference marks.

#2 DC2x Reference position establishment operation for a linear scale with reference marks is performed as follows:
0: The setting of bit 1 (DC4) of parameter No. 1802 is followed.
1: An absolute position is established by detecting two reference marks.

> **NOTE**
> 1 When this parameter is set to 1, specify the direction of the scale zero point by setting bit 4 (SCP) of parameter No. 1817.
> 2 When a rotary encoder with absolute address reference marks is used, this parameter is invalid. Even when this parameter is set to 1, the setting of bit 1 (DC4) of parameter No. 1802 is followed.

	#7	#6	#5	#4	#3	#2	#1	#0
1815						DCLx	OPTx	

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#1 OPTx The separate position detector is:
0: Not to be used (semi-closed system)
1: To be used (full-closed system)

> **NOTE**
> Set 1 when this function is used.

#2 DCLx As a separate position detector, a linear scale with reference marks or a linear scale with an absolute address zero point is:
0: Not used.
1: Used.

> **NOTE**
> Set 1 when this function is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1818						DG0		

[Input type] Parameter input
[Data type] Bit axis

#2 DG0 When the linear scale function with absolute address reference marks is used, reference position establishment operation based on the G00 command and jog feed is:
0: Disabled.
1: Enabled.

4.11 REFERENCE POSITION SIGNAL OUTPUT FUNCTION

Overview

If, after the establishment of the reference position on each axis, the coordinates in the machine coordinate system match the reference position, this function outputs a signal as the DO signal <Fn517> of the PMC.
Also, if the coordinates in the machine coordinate system match the second reference position, the function outputs a signal as the DO signal <Fn518> of the PMC.

Signal

Reference position match signals RP11 to RP18<Fn517>

[Classification] Output signal
[Function] Reports that the coordinates in the machine coordinate system match the reference position.
There is an independent one for each axis, with the last digit indicating the number of the corresponding controlled axis.
RP1x
x : 1 The machine coordinates on the first axis match the reference position.
2 The machine coordinates on the second axis match the reference position.
3 The machine coordinates on the third axis match the reference position.
: :
: :

[Output cond.] This signal becomes "1" only when, after the establishment of the reference position on each axis, the coordinates in the machine coordinate system match the reference position.

> **NOTE**
> This signal does not take the delay due to acceleration/deceleration into consideration.
> No in-position check is performed.

Second reference position match signals RP21 to RP28<Fn518>

[Classification] Output signal

[Function] Reports that the coordinates in the machine coordinate system match the second reference position.
There is an independent one for each axis, with the last digit indicating the number of the corresponding controlled axis.

RP2x

x : 1 The machine coordinates on the first axis match the second reference position.
2 The machine coordinates on the second axis match the second reference position.
3 The machine coordinates on the third axis match the second reference position.
: :
: :

[Output cond.] This signal becomes "1" only when, after the establishment of the reference position on each axis, the coordinates in the machine coordinate system match the second reference position.

> **NOTE**
> This signal does not take the delay due to acceleration/deceleration into consideration.
> No in-position check is performed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn517	RP18	RP17	RP16	RP15	RP14	RP13	RP12	RP11
Fn518	RP28	RP27	RP26	RP25	RP24	RP23	RP22	RP21

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1205			R2O	R1O				

[Input type] Parameter input
[Data type] Bit path

#4 R1O The output of the signal for the reference position is:
0: Disabled.
1: Enabled.

#5 R2O The output of the signal for the second reference position is:
0: Disabled.
1: Enabled.

NOTE

NOTE
1 For an axis subject to angular axis control, a signal is output in the angular coordinate system.
2 For the third/fourth reference positions and floating reference positions, no signals are output.

4.12 CORRESPONDENCE OF ROTARY SCALE WITHOUT ROTARY DATA

Overview

This manual describes how to deal with an absolute position detector (absolute pulse coder) or a scale with distance-coded reference marks (serial), when the rotary scale without rotary data is used, such as Heidenhain rotary scale RCN723 or Futaba rotary scale FRR902L3DB.
And it is required to set some servo parameters in order to use Heidenhain rotary scale RCN223, 723 and 220. In the case of using such a scale, please set servo parameters with referring to Subsection 4.14.5 “Method of using Heidenhain rotary scale RCN223, 723 and 220”.

⚠ **WARNING**
Please set the parameters that correspond to the absolute position detector which is actually used. If the parameter is not set correctly, the machine coordinates are not correctly established at power-on.

4.12.1 Setting Method by Rotary Axis Type and Movable Range

Rotary axis type	Movable range	Parameters required setting		Reference item
Rotary axis B type	Under one rotation	No.1817#3	No.1868	Subsec. 4.14.2
Rotary axis B type	Over one rotation	No.1815#0		Subsec. 4.14.3
Rotary axis A type	-	No.1815#6		Subsec. 4.14.4

See the following description for the rotary axial type.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx
#1 ROSx Setting linear or rotary axis.

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No. 1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

4.12.2 In the Case of a Rotary Axis B Type whose Movable Range is under One Rotation

In the case of a rotary axis B type whose movable range is under one rotation with an absolute position detector (absolute pulse coder) or a scale with distance-coded reference marks (serial), there is an uncontinuous point of scale data which is under one rotation, if a rotary scale without data (the number of rotation) is used. Then, if the scale data within the movable range of the rotary axis is continuous, it is possible to acquire the correct machine coordinate value of NC. On the other hand, if an uncontinuous point of scale data is in movable range, the machine coordinate value of NC is different from correct position.
In this function, by the bit 3 (SCRx) of parameter No.1817 and parameter No.1868, the machine coordinate value of NC become correct even if an uncontinuous point of scale data is in movable range. Therefore, it is possible to set a rotary axis B type even if there is no rotation speed data on the rotary scale on the rotary axis whose movable range is under one rotation.

Example 1

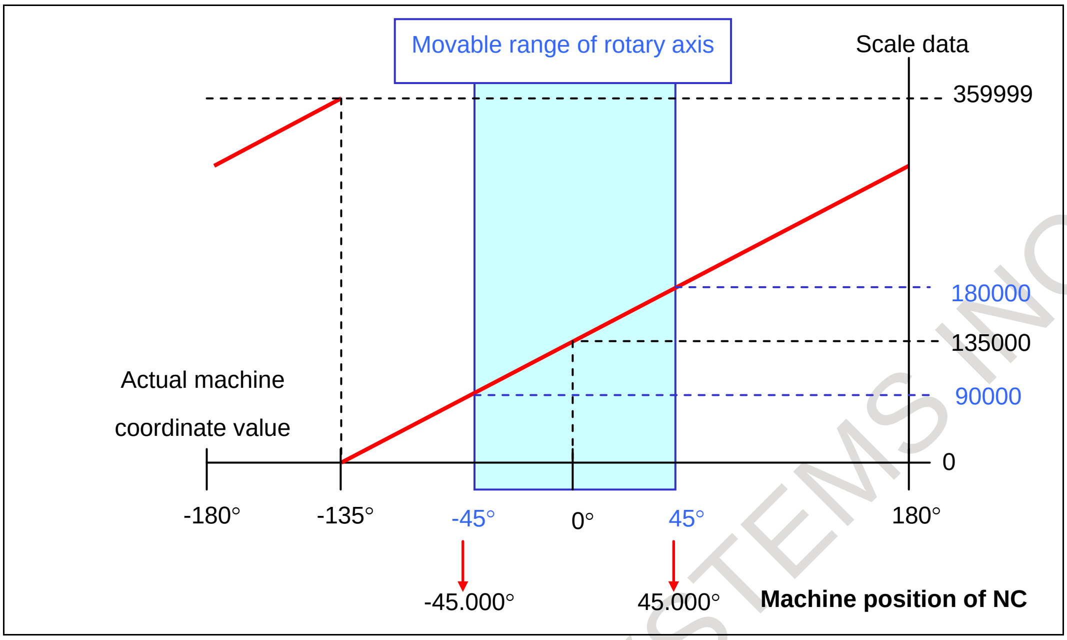


Fig. 4.12.2 (a) The case that the scale data is continuous in movable range of rotary axis

In the case of Fig. 4.12.2 (a), the scale data is continuous from 90000 to 18000.
Because the machine coordinate value of NC is shown by the following equation, (scale data when the power is turned on) - (scale data on the machine coordinate value equals 0), the machine coordinate value of NC becomes correct ones from -45.000 to 45.000.

Example 2

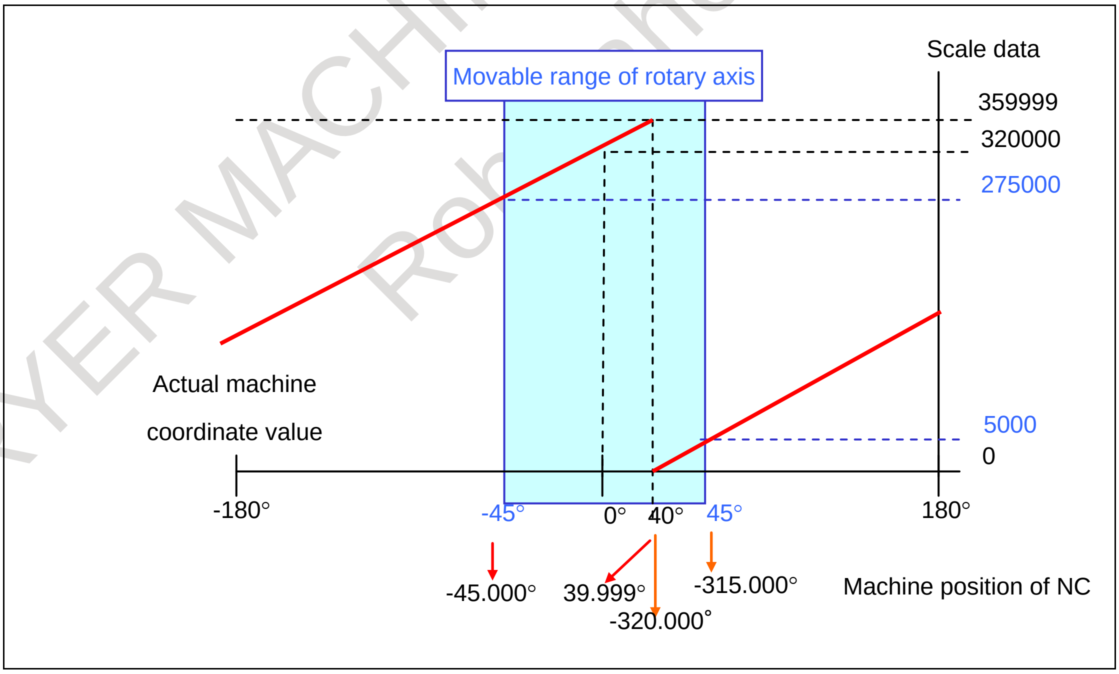


Fig. 4.12.2 (b) The case that the scale data is uncontinuous in movable range of rotary axis.

In the case of Fig. 4.12.2 (b), the scale data of movable range is uncontinuous, 275000 to 359999 and 0 to 5000, because the uncontinuous point whose machine coordinate value 40.000 exists.

Because the machine coordinate value of NC is shown by the following equation, (scale data when the power is turned on) - (scale data on the machine coordinate value equals 0),
NC shows two position ranges. One is correct ones from -45.000 to 0.000 to 39.999. And another is wrong ones from -320.000 to -315.000, which is different from correct position by 360 degree.

Example 3

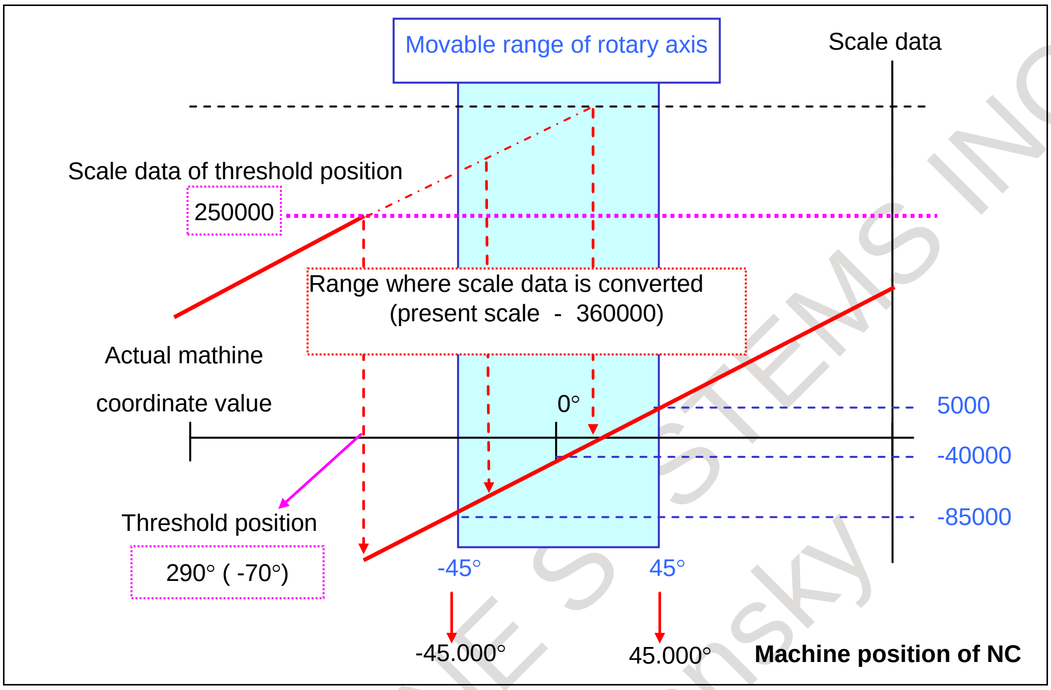


Fig. 4.12.2 (c) The case that parameter No.1817#3 is set to 1 and No.1868 (the threshold position) is set to 290.0.

In the case of Fig. 4.12.2 (b), if the bit 3 (SCRx) of parameter No. 1817 is set to 1 and No.1868 (threshold position) is set to 290.0, the scale data of the threshold position (parameter No.1868) or more (250000 to 359999) is converted to the data which is subtracted by the amount of one rotation (360000 in this case), as shown in Fig. 4.12.2 (c). Therefore, the scale data is continuous as -85000 to 5000 in movable range of the rotary axis. The machine coordinate value of NC can become correct value as -45.000 to 45.000 because the scale data is continuous in movable range of the rotary axis like Example 1.
The parameter No.1868 (the threshold position) is required to be set the position out of movable range and positive value. (0.0 to 360.0: Set to 290.0 instead of –70.0, in this case.)

An uncontinuous point of scale data

Before using this function, it is necessary to know the position of the uncontinuous point. By the following process, the position of the uncontinuous point can be found in the case of an absolute position detector (absolute pulse coder) (bit 5 (APC) of parameter No.1815=1).

1. Perform a reference position return of the rotary axis with the parameter No.1817#3=0.

→ The correspondence between the reference position and position in the absolute-position detector has been established. (parameter No.1815#4=1)

2. Record machine coordinate value after moving the axis to one side of end of movable range.

3. Turn the CNC off and on.

4. The machine coordinate value before and after the power supply turning on are compared.
- In the case that the machine coordinate value misses by the amount of one rotation :
  →Go to the process 7 because an uncontinuous point of scale data exists within movable range.
- In the case that the machine coordinate value does not miss and CNC starts normally :
  →Record the machine coordinate value after moving to the other side of end of movable range.

5. Turn the CNC off and on.

6. The machine coordinate value before and after the power supply turning on are compared.
- In the case that the machine coordinate value misses by the amount of one rotation :
  →Go to the process 7 because an uncontinuous point of scale data exists within movable range.
- In the case that the machine coordinate value does not miss and CNC starts normally :
  →There is not an uncontinuous point within movable range.

It is unnecessary to use this function.
(No.1817#3=0)

7. Record the machine coordinate value after moving by a constant interval (10 degree, for example) from present position for an inside of movable range.

8. Turn the CNC off and on.
- Until exceeding an uncontinuous point, the machine coordinate position is still different by 360 degree.
  → Until exceeding an uncontinuous point, repeat process 7 and 8.
- After exceeding an uncontinuous point, the CNC starts on correct position.
  →By the above process it is possible to know the approximate position of the uncontinuous point.

In the case of the Fig. 4.12.2 (b) :

1. A reference position return finishing.
2. Moving the axis to one side of movable range (the machine coordinate value -45.000).
3. Turnning the CNC off and on.
4. Moving to the other side of movable range (the machine coordinate value is 45.000) because the machine coordinate value at power on is correct.
5. Turnning the CNC off and on.
6. Starting on -315.000 with the machine coordinate missing by 360 degree.
7. Shifting to the inside direction by 10 degree and moving to the machine coordinate -325.000.
8. Turnning the CNC off and on.

9. Starting correctly on the machine coordinate 35.000.

Thus an uncontinuous point of scale data is found between 35.000 and 45.000.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx	APZx				

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#4 APZx Machine position and position on absolute position detector when the absolute position detector is used
0: Not corresponding
1: Corresponding
When an absolute position detector is used, after primary adjustment is performed or after the absolute position detector is replaced, this parameter must be set to 0, power must be turned off and on, then manual reference position return must be performed. This completes the positional correspondence between the machine position and the position on the absolute position detector, and sets this parameter to 1 automatically.

#5 APCx Position detector
0: Other than absolute position detector
1: Absolute position detector (absolute Pulsecoder)

> **NOTE**
> 1 In case of using FANUC absolute pulsecoder as absolute position detector for the rotary axis A type that machine coordinate values are rounded in 0 to 360°, set the parameter RON (No.1815#6) to 0. If the parameter is not set correctly, the machine coordinates are not correctly established at power-on.
> 2 In case of using a rotary scale without rotary data made by another company as absolute position detector, please refer to "CORRESPONDENCE OF ROTARY SCALE WITHOUT ROTARY DATA" in the “CONNECTION MANUAL (FUNCTION)” (B-64693EN-1) and set an appropriate value corresponding to the detector. If the parameter is not set correctly, the machine coordinates are not correctly established at power-on.
> 3 When the servo axis is disabled (bit 4 (KSVx) of parameter No. 11802 is set to 1), an absolute position detector cannot be used (bit 5 (APCx) of parameter No. 1815 cannot be set to 1). If an absolute position detector is used, alarm SV0301, "APC ALARM; COMMUNICATION ERROR" is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
1817					SCRx			

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#3 SCRx Specifies whether to convert scale data by using threshold position (parameter No. 1868) so that rotary axis B type is available, in the case of the axis B type that use a rotary scale without data (the number of rotation), whose movable range is under one rotation:
0 : Not to convert.
1 : To convert.

> **NOTE**
> 1 This parameter is available for only the rotary axis B type with an absolute position detector (absolute Pulsecoder) or a rotary scale with distance-coded reference marks (serial). This function cannot be used for distance coded rotary scale interface (phase A/B).
> 2 Don't set this parameter in the case of no uncontinuous point within movable range of rotary axis even if the rotary axis B type.
> 3 When this parameter is set, machine position and position on absolute position detector become uncorresponding. So, establish the reference position again. (Consequently, the bit 4 (APZ) of parameter No. 1815 (indicating that the correspondence is established) is set to 0, and an alarm DS0300, "APC ALARM: NEED REF RETURN", occurs. Why the bit 4 (APZ) of parameter No. 1815 is set to 0 can be checked using diagnosis data No. 310#0).
> 4 This function cannot be used together with the bit 0 (RVSx) of parameter No. 1815 that save rotary data by CNC.
> 5 In this function, the amount of one rotation of rotary axis assumes 360, and the machine position 0 assumes the reference position. It is not possible to apply to a rotary axis other than the above-mentioned setting.
> 6 Set the parameter No. 1240 to 0.

1868	Threshold position for converting scale data (each axis)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (Refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to ++999999.999)

In the case that scale data of a rotary scale without rotary data is larger than the scale data of the threshold position (this parameter value), it is converted to be continuous data in movable range by subtracting data of one rotation. The machine coordinate value out of movable range (angle from an uncontinuous point) must be set as threshold position. As for the axis for which this parameter is set to 0, conversion of scale data is not performed.

> **NOTE**
> 1 This parameter is available for only the rotary axis B type with an absolute position detector (absolute Pulsecoder) or a rotary scale with distance-coded reference marks (serial), for which the bit 3 (SCRx) of parameter No. 1817 is set to 1.
> 2 This function cannot be used for distance coded rotary scale interface (phase A/B).
> 3 Don't set this parameter in the case of no uncontinuous point within movable range of rotary axis even if the rotary axis B type.
> 4 When this parameter is set, machine position and position on absolute position detector become uncorresponding. So, establish the reference position again. (Consequently, the bit 4 (APZ) of parameter No. 1815 (indicating that the correspondence is established) is set to 0, and an alarm DS0300, "APC ALARM: NEED REF RETURN", occurs. Why the bit 4 (APZ) of parameter No. 1815 is set to 0 can be checked using diagnosis data No. 0310#0).

1869	The amount of one rotation of rotary axis B type (each axis)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (Refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to ++999999.999)
Normally, the amount of one rotation of rotary axis is 360, and the machine position 0 is the reference position.
In this case, this parameter is set to 0.

For instance, when this parameter is set to 523.000, the amount of one rotation become 523.000 (in the case of IS-B), if it is necessary to set it arbitrarily.

> **NOTE**
> 1 This parameter is available for only the rotary axis B type with an absolute position detector (absolute Pulsecoder) or a rotary scale with distance-coded reference marks (serial), as for the bit 3 (SCRx) of parameter No. 1817 is set to 1 or the bit 0 (RVS) of parameter No. 1815 is set to 1.

> **NOTE**
> 2 In the case that the amount of one rotation of rotary axis is 360, this parameter is set to 0.If it is necessary to set an amount of one rotation of rotary axis arbitrarily, this parameter is set to the amount of one rotation.
> 3 When this parameter is set, machine position and position on absolute position detector become uncorresponding. So, establish the reference position again. (Consequently, the bit 4 (APZ) of parameter No. 1815 (indicating that the correspondence is established) is set to 0, and an alarm DS0300, “APC ALARM: NEED REF RETURN”, occurs. Why the bit 4 (APZ) of parameter No. 1815 is set to 0 can be checked using diagnosis data No. 0310#0).
> 4 This parameter No. 1869 is common in movable range that is under one rotation (the bit 3 (SCRx) of parameter No. 1817 is set to 1) and movable range that is over one rotation (the bit 0 (RVS) of parameter No. 1815 is set to 1).
> 5 Please set 0 to this parameter if the distance coded rotary scale interface (phase A/B) is used.

4.12.3 In the Case of a Rotary Axis B Type whose Movable Range is over One Rotation

As for an axis whose movable range is over one rotation and its rotary scale which has no rotary data , for example, 30.000 degree and 390.000 degree have completely same scale data though their position is different from each other by just one rotation. It is possible to set rotary axis B type because CNC saves the rotary data by the following parameter even if in this case.
As attention, a machine coordinate value may get out over a rotation, in the case of moving over 180 degree during turning off, because CNC saves a machine coordinate value just before CNC turns off and in following turning on get from the value.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1815								RVSx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 RVSx When the scale without rotary data is used:
0: The CNC does not maintain the rotary data.
1: The CNC maintains the rotary data.
This parameter is effective in the axis that satisfies either of the following conditions.
- The axis is rotary axis B type and the movable range is more than one rotation.
- The parameter RVL(No.11802#6)=1.

NOTE

1. In the case of a rotary axis B type whose movable range is over one rotation, a rotary scale with rotary data had better be used. Otherwise, a machine coordinate value may get out over a rotation, in the case of moving over 180 degree during turning off.
2. This parameter is available for the axis that is rotary axis B type or the axis whose bit 6 (RVLx) of parameter No.11802 is 1. This parameter is available for the axis with an absolute position detector (absolute Pulsecoder), a distance coded rotary scale interface (phase A/B) or a rotary scale with distance-coded reference marks (serial).
3. If this parameter is available, the machine coordinate value just before CNC turns off is saved. In the case of moving over the following amounts during turning off or before reference position return, a machine coordinate value may get out over a rotation because CNC saves a machine coordinate value just before CNC turns off and in following turning on get from the value.
   In case of the rotary axis B type :
   Over 180 degree
   In case of the linear axis type :
   Over 1/4 of values set to parameter No.11810.
4. If this parameter is changed, the correspondence between the machine position and the absolute position detector is lost. So, establish the reference position again. (Bit 4 (APZ) of parameter No.1815 is set to 0, and an alarm DS0300, “APC ALARM: NEED REF RETURN” is issued. The cause that sets bit 4 (APZ) of parameter No.1815 to 0 is indicated in diagnosis data No.310#0).
5. Absolute coordinate value is set by machine coordinate value. However, after CNC turns on, the workpiece offset such as G92 and G52 executed before CNC turns off is not set.
6. This function cannot be used together with the bit 3 (SCRx) of parameter No. 1817 that convert scale data.
7. In case of the rotary axis B type and the amount of one rotation of rotary axis is 360, set the parameter No.1869 to 0. Moreover, set the parameter No.1240 to 0 because the reference position must be 0 degree.
8. In case of the rotary axis B type and the amount of one rotation of rotary axis is not 360, set the parameter No.1869 to the amount of one rotation. Moreover, set the parameter No.1240 to 0 because the reference position must be 0 degree.
9. In case of the linear axis type, set the amount of the movement per one motor rotation to the parameter No.11810.
10. This parameter cannot be used together with the stored stroke limit check immediately after power-on (parameter DOT(No.1311#0)).

1869	**The amount of one rotation of rotary axis B type (each axis)**

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (Refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to ++999999.999)
Normally, the amount of one rotation of rotary axis is 360, and the machine position 0 is the reference position.
In this case, this parameter is set to 0.

For instance, when this parameter is set to 523.000, the amount of one rotation become 523.000 (in the case of IS-B), if it is necessary to set it arbitrarily.

> **NOTE**
> 1 This parameter is available for only the rotary axis B type with an absolute position detector (absolute Pulsecoder) or a rotary scale with distance-coded reference marks (serial), as for the bit 3 (SCRx) of parameter No. 1817 is set to 1 or the bit 0 (RVS) of parameter No. 1815 is set to 1.
> 2 In the case that the amount of one rotation of rotary axis is 360, this parameter is set to 0.If it is necessary to set an amount of one rotation of rotary axis arbitrarily, this parameter is set to the amount of one rotation.
> 3 When this parameter is set, machine position and position on absolute position detector become uncorresponding. So, establish the reference position again. (Consequently, the bit 4 (APZ) of parameter No. 1815 (indicating that the correspondence is established) is set to 0, and an alarm DS0300, “APC ALARM: NEED REF RETURN”, occurs. Why the bit 4 (APZ) of parameter No. 1815 is set to 0 can be checked using diagnosis data No. 0310#0).
> 4 This parameter No. 1869 is common in movable range that is under one rotation (the bit 3 (SCRx) of parameter No. 1817 is set to 1) and movable range that is over one rotation (the bit 0 (RVS) of parameter No. 1815 is set to 1).
> 5 Please set 0 to this parameter if the distance coded rotary scale interface (phase A/B) is used.

4.12.4 In the Case of a Rotary Axis A Type

In the case of a rotary axis A type with an absolute position detector (absolute pulse coder), when the machine coordinate passes 0 degree or the shift value per one rotation (360 degree or the value of parameter No.1260), the data for the reference position (parameters Nos.1860 to 1862) is renewed so that following up at power on is within one rotation. But as for the absolute position detector (absolute pulse

coder) using a scale without rotary data, the data for the reference position (parameters Nos.1860 to 1862) need not be renewed because scale data without rotary data is always read at turning on.
In this case, be sure to set the following parameters.

Parameter

1260	The shift amount per one rotation of a rotary axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the shift amount per one rotation of a rotary axis.
For the rotary axis used for cylindrical interpolation, set the standard value.

	#7	#6	#5	#4	#3	#2	#1	#0
1815		RONx						

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#6 RONx With a rotary axis A type, an absolute position detector (absolute Pulsecoder) using a scale without rotary data is:
0: Not used.
1: Used.

> **NOTE**
> 1 This parameter is available for only the rotary axis A type with an absolute position detector (absolute Pulsecoder). This function cannot be used for a rotary scale with distance-coded reference marks (serial) or for a distance coded rotary scale interface (phase A/B).
> 2 Set this parameter to 1 for the rotary axis A type using a rotary scale without rotary data such as HEIDENHAIN rotary scale RCN 223F, 727F, etc.
> Set this parameter to 0 in any other case. If the parameter is not set correctly, the machine coordinates are not correctly established at power-on.

NOTE

3 When this parameter is set, machine position and position on absolute position detector become uncorresponding. So, establish the reference position again. (Consequently, the bit 4 (APZ) of parameter No. 1815 (indicating that the correspondence is established) is set to 0, and an alarm DS0300, “APC ALARM: NEED REF RETURN”, occurs. Why the bit 4 (APZ) of parameter No. 1815 is set to 0 can be checked using diagnosis data No. 0310#0).

1860	**Value 1 for the zero point of the absolute position detector**
1861	**Value 2 for the zero point of the absolute position detector**
1862	**Value 3 for the zero point of the absolute position detector**

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Locked parameter
[Data type] 2-word axis

Parameters Nos.1860 to 1862 are values for the zero point of the absolute position detector (association with reference position and the counter value of the absolute position detector).
The CNC automatically sets parameters Nos.1860 to 1862 when correspondence between the reference position and the absolute position detector is performed.
It is not available to set parameters Nos.1860 to 1862 by using inputting of MDI operation, FOCAS2, PMC window or programmable parameter input(G10).

The zero point data of the absolute position detection can be restored and parameters Nos.1860 to 1862 can be set, by the parameter input from connected I/O unit or by restoring the SRAM area of boot system.
When the zero point data of the absolute position detection are restored, be careful about the following.

- When the zero point data of absolute position detection are restored, back up them right before the restoring and use backup data.
- When the motor or the detector is exchanged, the zero point data of absolute position detection cannot be restored. After exchange, set the zero point of the absolute position detector again.
- When the zero point data of the absolute position detection are restored by them of the other machine, the zero point of the absolute position detector is set to wrong position. Don't restore the zero point data of absolute position detection by them of the other machine.
- When the zero point data of absolute position detection are backed up with reference position not established and restored by backup data, setting of the zero point of the absolute position detector is required after restoring.

⚠ **WARNING**

If the zero point of the absolute position detector is not set properly, the machine coordinate system is not established correctly.

4.12.5 Method of Using Heidenhain Rotary Scale RCN223, 723 and 220

Heidenhain rotary scale RCN223, 723 and 220 are the detectors which save absolute position only within one rotation.

- In the case of RCN223, 723 and 220, there is the case that wrong positional data (rotary data) is read on absolute positional communication because it is not saved (it is undefined).

And the former two detectors (RCN223 and 723) are the detectors which have eight million pulses every one rotation.

- In the case of RCN223 and 723, if it is used as usual, eight grids are made every one rotation.

The following parameters in addition to "setting method by rotary axis type and movable range" (Subsection 4.14.1 - 4.14.4) are necessary to resolve these problems.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
2275							RCNCLR	800PLS

[Input type] Parameter input
[Data type] Bit axis

#0 800PLS A rotary encoder (RCN223, 723 or etc.) with eight million pulses per revolution is:
0: Not to be used. (To use the RCN220, leave this bit set to 0.)
1: To be used. (1 grid point occur per revolution.)

#1 RCNCLR The number of revolution is:
0: Not to be cleared.
1: To be cleared. (To use the RCN220, RCN223, or RCN723, set the bit to 1.)
This function bit is to be set in combination with the number of data mask digits, described in parameter (No.2394).

2394	Number of data mask digits

[Input type] Parameter input
[Data type]
[Setting value] 5 or 8
The value to be set in this parameter depends on the detector. At present, only the following detectors require clearing the speed data. Set 5 to use the RCN220, and set 8 to use the RCN223, or RCN723. This parameter is to be set in combination with RCNCLR(No.2275#1).

NOTE
The rotary data of the RCN220, RCN223, or RCN723 is maintained while the power to the separate detector interface unit is on. The data, however, is cleared when the unit is turned off.
Since the rotary data becomes undetermined depending on where the power is turned off, it is necessary to make a setting to clear the rotary data. In addition, for this reason, the RCN220,RCN223, and RCN723 cannot be used with a linear axis.
However, the detectors are available with a linear axis even if the rotary data is cleared by setting the parameters RVL(No.11802 #6)=1 and RVS(No.1815#0)=1.

- Other parameters that must be set

When using RCN220, the other settings are the same as normal α/αi pulse coders. The setting method for RCN223 and RCN723 are as follows.

[Flexible feed gear]
Parameters Nos.2084, 2085
In the case of RCN223, 723, amount of pulses per every one rotation are regarded as eight million pulses.

$$\text{<flexible feed gear (N/M)>} = \frac{\dfrac{\text{Amount of table moving every one rotation of a detector [deg]}}{\text{Increment of detection [deg]}}}{8{,}000{,}000}$$

[Amount of positional pulse]
Parameter No.2024
amount of positional pulse = 100,000 * (deceleration ratio from a motor to a table)

* As a result of this equation when amount of pulses exceeds input range (0 to 32767), set the positional pulse in following process with using ”positional feed back pulse conversion coefficient”
Amount of positional pulse = A × B
In this case, select B in which A is within 32767. And A, B are set as following.
A : A parameter, amount of positional pulse (within 32767)
No.2024
B : A parameter, positional feed back pulse conversion coefficient
No.2185

[Reference counter capacity]
Parameter No.1821
Set amount of feed back pulses (detection unit) per a table rotation.

* Set amount of (pulses per one table rotation / 8) as reference counter capacity in the case of bit 0 (800PLS) of parameter No. 2275=0. But in this case eight grids are made every one table rotation.

- Example of parameters setting

[System construction]
- Heidenhain rotary encorder RCN223
- The least input increment is 1/10,000 degree
- In the case that moving distance is 180 degree every one motor rotation (deceleration ratio is 1/2)
- Deceleration ratio between a rotary table and a separate encorder = 1/1

[Setting parameters]
- Set bit 1 (OPTx) of parameter No. 1815=1 so that separate detector is available.
- Set bit 0 (800PLS) of parameter No. 2275=1, bit 1 (RCNCLR) of parameter No. 2275=1, No.2394=8 so that RCN223 is used.
- Parameters for flexible feed gear are calculated.
  Flexible feed gear (N/M) = (360 degree / 0.0001 degree) / 8,000,000 =9 / 20
  So, No.2084=9, No.2085=20
- Amount of positional pulses is calculated.
  Amount of positional pulses = 100,000 × (1 / 2) = 50,000
  Because this value exceeds input range (0 to 32767), following is considered
  50,000 = 12,500 × 4 → A = 12,500, B=4
  A is set for “amount of positional pulse” and B is set for “positional pulse conversion coefficient”
  No.2024＝12,500, No.2185=4
- Reference counter capacity is calculated.
  Reference counter capacity = 360 degree / 0.0001 degree = 3,600,000

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1815							OPTx	

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#1 OPTx The separate position detector is:
0: Not to be used (semi-closed system)
1: To be used (full-closed system)

> **NOTE**
> 1 In case of using the absolute position detector (bit 5 (APCx) of parameter No.1815 is set to 1), please set the following parameters that correspond to the absolute position detector which is actually used.
> - parameter No.1815#6, No.1815#0, No.1817#3, No.1868, No.2275#1, No.2394
>
> If these parameters are not set correctly, the machine coordinates are not correctly established at power-on.
> 2 Set this parameter to 1 when using a linear scale with reference marks or a linear scale with distance-coded reference marks (serial) (full-closed system).

1821	Reference counter size for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

Set a reference counter size.
As a reference counter size, specify a grid interval for reference position return based on the grid method.
When a value less than 0 is set, the specification of 10000 is assumed.
When a linear scale with absolute address reference marks is used, set the interval of mark 1.

2024	Amount of positional pulse

[Input type] Parameter input
Set an amount of positional pulse.
amount of positional pulse = 100,000 * (deceleration ratio from a motor to a table)

As a result of this equation when amount of pulses exceeds input range (0 to 32767), set the positional pulse in following process with using "positional feed back pulse conversion coefficient"

Amount of positional pulse = A × B

In this case, select B in which A is within 32767. And A, B are set as following.

A : A parameter, amount of positional pulse (within 32767)
No.2024

B : A parameter, positional feed back pulse conversion coefficient
No.2185

2084	Flexible feed gear (numerator)

2085	Flexible feed gear (denominator)

[Input type] Parameter input

Set a flexible feed gear.

In the case of RCN223, 723, amount of pulses per every one rotation are regarded as eight million pulses.

<flexible feed gear (N/M)> =

$$\frac{\dfrac{\text{Amount of table moving every one rotation of a detector [deg]}}{\text{Increment of detection [deg]}}}{8{,}000{,}000}$$

2185	Positional feed back pulse conversion coefficient

[Input type] Parameter input

Set a positional feed back pulse conversion coefficient.

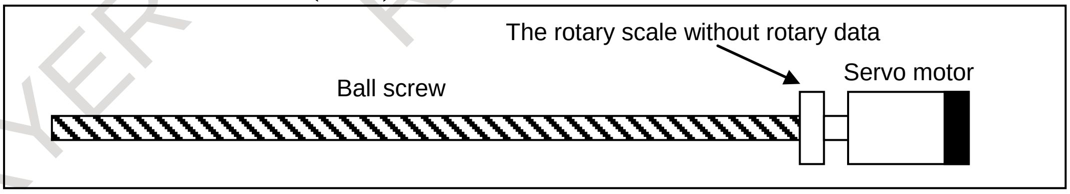
Use this parameter when amount of positional pulses (the parameter No.2024) exceeds input range (0 to 32767).

Refer to the parameter No.2024 for the setting method.

4.12.6 In Case of a Linear Axis

Explanation

An absolute position detection using a rotary scale without rotary data or a linear scale with distance-coded reference marks (serial) is available.



This parameter is effective in the axis that satisfies all the following conditions.

- Parameter ROT(No.1006#0)=0
- Parameter ROS(No.1006#1)=0
- Parameter RVS(No.1815#0)=1
- Parameter RVL(No.11802#6)=1
- The amount of the movement per rotation is set to parameter No.11810.

NOTE

1 To use this function, please set the parameters that relate to the rotary scale without rotary data correctly : 800PLS, RCNCLR(No.2275#0 and #1), and parameter No.2394. About details, please refer to the "4.14.5 Method of Using Heidenhain Rotary Scale RCN223, 723 and 220".

> **NOTE**
> 2 To change this function from effective to ineffective, set the related parameters again correctly.
> 3 Please refer also to notes that described in parameters RVS(No.1815#0) and No.11810.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006							**ROSx**	**ROTx**

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx
#1 ROSx Setting linear or rotary axis.

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No. 1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

	#7	#6	#5	#4	#3	#2	#1	#0
1815								**RVSx**

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 RVSx When the scale without rotary data is used:

0: The CNC does not maintain the rotary data.

1: The CNC maintains the rotary data.

This parameter is effective in the axis that satisfies either of the following conditions.

- The axis is rotary axis B type and the movable range is more than one rotation.
- The parameter RVL(No.11802#6)=1.

> **NOTE**
> 1 In the case of a rotary axis B type whose movable range is over one rotation, a rotary scale with rotary data had better be used.
> 2 This parameter is available for the axis that is rotary axis B type or the axis whose bit 6 (RVLx) of parameter No.11802 is 1. This parameter is available for the axis with an absolute position detector (absolute Pulsecoder), a distance coded rotary scale interface (phase A/B) or a rotary scale with distance-coded reference marks (serial).
> 3 If this parameter is available, the machine coordinate value just before CNC turns off is saved. In the case of moving over the following amounts during turning off or before reference position return, a machine coordinate value may get out over a rotation because CNC saves a machine coordinate value just before CNC turns off and in following turning on get from the value.
> In case of the rotary axis B type :
> Over 180 degree
> In case of the linear axis type :
> Over 1/4 of values set to parameter No.11810.
> 4 If this parameter is changed, the correspondence between the machine position and the absolute position detector is lost. So, establish the reference position again. (Bit 4 (APZ) of parameter No.1815 is set to 0, and an alarm DS0300, “APC ALARM: NEED REF RETURN” is issued. The cause that sets bit 4 (APZ) of parameter No.1815 to 0 is indicated in diagnosis data No.310#0).
> 5 Absolute coordinate value is set by machine coordinate value. However, after CNC turns on, the workpiece offset such as G92 and G52 executed before CNC turns off is not set.
> 6 This function cannot be used together with the bit 3 (SCRx) of parameter No. 1817 that convert scale data.
> 7 In case of the rotary axis B type and the amount of one rotation of rotary axis is 360, set the parameter No.1869 to 0. Moreover, set the parameter No.1240 to 0 because the reference position must be 0 degree.
> 8 In case of the rotary axis B type and the amount of one rotation of rotary axis is not 360, set the parameter No.1869 to the amount of one rotation. Moreover, set the parameter No.1240 to 0 because the reference position must be 0 degree.
> 9 In case of the linear axis type, set the amount of the movement per one motor rotation to the parameter No.11810.

> **NOTE**
> 10 This parameter cannot be used together with the stored stroke limit check immediately after power-on (parameter DOT(No.1311#0)).

	#7	#6	#5	#4	#3	#2	#1	#0
2275							RCNCLR	800PLS

[Input type] Parameter input
[Data type] Bit axis

#0 800PLS A rotary encoder (RCN223, 723 or etc.) with eight million pulses per revolution is:
0: Not to be used. (To use the RCN220, leave this bit set to 0.)
1: To be used. (1 grid point occur per revolution.)

#1 RCNCLR The number of revolution is:
0: Not to be cleared.
1: To be cleared. (To use the RCN220, RCN223, or RCN723, set the bit to 1.)
This function bit is to be set in combination with the number of data mask digits, described in parameter (No.2394).

2394	Number of data mask digits

[Input type] Parameter input
[Data type]
[Setting value] 5 or 8
The value to be set in this parameter depends on the detector. At present, only the following detectors require clearing the speed data. Set 5 to use the RCN220, and set 8 to use the RCN223, or RCN723. This parameter is to be set in combination with RCNCLR(No.2275#1).

> **NOTE**
> The rotary data of the RCN220, RCN223, or RCN723 is maintained while the power to the separate detector interface unit is on. The data, however, is cleared when the unit is turned off.
> Since the rotary data becomes undetermined depending on where the power is turned off, it is necessary to make a setting to clear the rotary data. In addition, for this reason, the RCN220,RCN223, and RCN723 cannot be used with a linear axis.
> However, the detectors are available with a linear axis even if the rotary data is cleared by setting the parameters RVL(No.11802 #6)=1 and RVS(No.1815#0)=1.

	#7	#6	#5	#4	#3	#2	#1	#0
11802		RVL						

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#6 RVL In case of using the rotary scale without rotary data to the linear axis type, an absolute position detector or a rotary scale with distance-coded reference marks (serial) is:
0: Not available.
1: Available.

> **NOTE**
> 1 Please use this parameter with a linear axis.
> 2 This parameter is effective when the parameter RVS(No.1815#0) is 1.
> 3 Set the parameter No.11810 to the amount of one rotation.

11810	The amount of the movement per one motor rotation of linear axis type (each axis)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
In case of using the rotary scale without rotary data to the linear axis type, set the amount of the movement per one motor rotation. When 0 is set in this parameter, 360. is assumed.

> **NOTE**
> 1 This parameter is effective in the axis that satisfies all the following conditions.
> - Linear axis
> - Parameter RVS(No.1815#0)=1
> - Parameter RVL(No.11802#6)=1
> 2 This parameter is available for the axis with an absolute position detector (absolute Pulsecoder) or a rotary scale with distance-coded reference marks (serial).
> 3 If this parameter is changed, the correspondence between the machine position and the absolute position detector is lost. So, establish the reference position again. (Bit 4 (APZ) of parameter No.1815 is set to 0, and an alarm DS0300, "APC ALARM: NEED REF RETURN" is issued. The cause that sets bit 4 (APZ) of parameter No.1815 to 0 is indicated in diagnosis data No.310#0).

4.12.7 Diagnosis Data

This diagnosis data shows the reason that the parameter APZ is set to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
DGN 310								**PR1**

#0 PR1 One of the following parameters was changed:
No.1815#0, No.1815#1, No.1815#6, No.1817#3, No.1820, No.1821, No.1822, No.1823, No.1850, No.1868, No.1869, No.1874, No.1875, No.1876, No.1878, No.1883, No.1884, No.2022, No.2084, No.2085, No.2179, increment system for a rotary axis A type, No.11802#6, No.11810

4.13 MANUAL 2ND/3RD/4TH REFERENCE POSITION RETURN FUNCTION

Overview

This function enables positioning to the 2nd/3rd/4th reference position by JOG feed operation in manual reference position return mode.

Use the manual 2nd/3rd/4th reference position return select 1 signal SLREF <Gn340.5> and the manual 2nd/3rd/4th reference position return select 2 signal SLRER <Gn340.6> to select the reference position to which to return, then set the feed axis direction selection signal +/-Jx to 1 in manual reference position return mode to enable 2nd/3rd/4th reference position return.

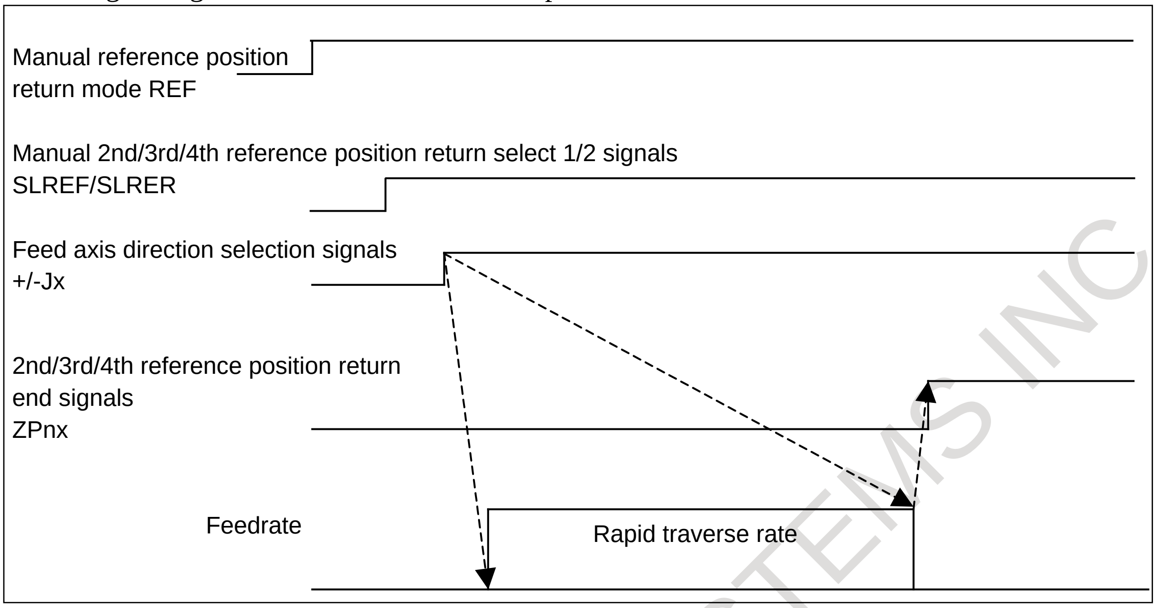
The feedrate is set by rapid traverse rate parameter No. 1420 for each axis. (Rapid traverse override is effective.)
By using bit 2 (JZR) of parameter No. 1401, the jog feedfrate can be used for manual reference position return at all time.
Bit 1 (DLF) of parameter No. 1404 can be used to select rapid traverse rate parameter No. 1420 for each axis or manual rapid traverse rate parameter No. 1424. This parameter selects a feedrate for reference position return performed without dogs. This parameter also selects a feedrate when manual reference position return is performed according to bit 7 (SJZ) of parameter No.0002 using rapid traverse without deceleration dogs after a reference position is set.

When the tool returns to the reference position of a controlled axis, the following reference position return end signal is set to 1.

(ZP2x <Fn096> for return to the 2nd reference position, ZP3x <Fn098> for return to the 3rd reference position, or ZP4x <Fn100> for return to the 4th reference position)

Although one axis can be moved at the same time, using bit 0 (JAX) of parameter No. 1002 allows up to three axes to move at the same time.
The 2nd, 3rd, and 4th reference positions must be set in advance in parameters Nos. 1241 to 1243 with coordinates in the machine coordinate system.
2nd/3rd/4th reference position return can be used only after the relevant reference position has been established.

The following timing chart summarizes the above procedure:



Caution

⚠ CAUTION

1 When high-speed manual reference position return is used, selecting a feed axis direction selection signal in manual reference position return mode may position the tool at the reference position irrespective of the direction specified by the feed axis direction selection signal, depending on the current position.

2 For manual 1st reference position return, be sure to set the manual 2nd/3rd/4th reference position return select 1/2 signals SLREF <Gn340.5> and SLRER <Gn340.6> to 0.

Note

NOTE

1 Manually returning to the 2nd reference position requires for manual 2nd/3rd/4th reference position return.
Also, manually returning to the 3rd or 4th reference position requires for manual 2nd/3rd/4th reference position return and 3rd/4th reference position return.

2 Once the reference position return end signal has been set to 1, the machine cannot run again in JOG mode until it is released from the reference position return mode.

3 The reference position return end signal will be set to 0 in the following cases:
- The tool moved from the reference position.
- Emergency stop is applied.
- A servo alarm is raised.
- The servo is turned off.

4 Do not change the manual 2nd/3rd/4th reference position return select 1/2 signal during moving for reference position return. Even if the signal is changed, the tool returns to the reference position selected at the start.

NOTE

5 When 1st reference position return has never be performed (the 1st reference position has not been established), returning to the 2nd/3rd/4th reference position is not executed.
6 This function cannot be used in composite control or superimposed control mode.
7 This function cannot be used for the Cs contour control axis.
8 For details of the mode change to the manual reference position return mode, refer to the “MODE SELECTION” in this manual.
9 For details of the high-speed manual reference position return, refer to the “MANUAL REFERENCE POSITION RETURN” in this manual.

Signal

Manual 2nd/3rd/4th reference position return select 1 signal SLREF <Gn340.5>
Manual 2nd/3rd/4th reference position return select 2 signal SLRER <Gn340.6>

[Classification] Input signal

[Function] These signals select the 1st, 2nd, 3rd, or 4th reference position to which to return the tool using the manual reference position return function.

SLRER <Gn340.6>	SLREF <Gn340.5>	
0	0	1st reference position return
0	1	2nd reference position return
1	0	3rd reference position return
1	1	4th reference position return

2nd reference position return end signals ZP21 to ZP28 <Fn096>
3rd reference position return end signals ZP31 to ZP38 <Fn098>
4th reference position return end signals ZP41 to ZP48 <Fn100>

[Classification] Output signal

[Function] These signals indicate that the tool currently exists at the 2nd, 3rd, or 4th reference position on a controlled axis.
The signals are available to each axis; the numeric character at the end of each signal name represents the number of a controlled axis and the one immediately following ZP represents the number of a reference position.

ZP x y

x : 2 ..... 2nd reference position return
3 ..... 3rd reference position return
4 ..... 4th reference position return

y : 1 ..... Return end signal for the 1st axis
2 ..... Return end signal for the 2nd axis
3 ..... Return end signal for the 3rd axis
: :

[Output cond.] The output is 1 when:

- The tool has returned to the 2nd, 3rd, or 4th reference position and is in the in-position state.

The output is 0 when:

- The tool moved from the reference position.
- Emergency stop is applied.
- A servo alarm is raised.
- The servo is turned off.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn340		SLRER	SLREF					
Fn096	ZP28	ZP27	ZP26	ZP25	ZP24	ZP23	ZP22	ZP21
Fn098	ZP38	ZP37	ZP36	ZP35	ZP34	ZP33	ZP32	ZP31
Fn100	ZP48	ZP47	ZP46	ZP45	ZP44	ZP43	ZP42	ZP41

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0002	SJZ							

[Input type] Setting input
[Data type] Bit

#7 SJZ On an axis for which bit 3 (HJZx) of parameter No. 1005 is set:
- 0: If a reference position is not established yet, reference position return is performed with deceleration dogs.
  If a reference position is already established, reference position return is performed at a parameter-set feedrate without using deceleration dogs.
- 1: Reference position return is performed with deceleration dogs at all times.

> **NOTE**
> SJZ is valid for an axis for which bit 3 (HJZx) of parameter No. 1005 is set to 1. When bit 1 (DLZx) of parameter No. 1005 is set to 1, however, manual reference position return after a reference position is set is performed at a parameter-set feedrate, regardless of the setting of SJZ.

	#7	#6	#5	#4	#3	#2	#1	#0
1002								JAX

[Input type] Parameter input
[Data type] Bit path

#0 JAX Number of axes controlled simultaneously in jog feed, manual rapid traverse and manual reference position return
- 0: 1 axis
- 1: 3 axes

1241	Coordinate value of the second reference position in the machine coordinate system
1242	Coordinate value of the third reference position in the machine coordinate system
1243	Coordinate value of the fourth reference position in the machine coordinate system

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg(machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the coordinate values of the second to fourth reference positions in the machine coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
1401						JZR		

[Input type] Parameter input
[Data type] Bit path

#2 JZR The manual reference position return at jog feedrate
0: Not performed
1: Performed

	#7	#6	#5	#4	#3	#2	#1	#0
1404							DLF	

[Input type] Parameter input
[Data type] Bit path

#1 DLF After a reference position is set, manual reference position return performed at:
0: Rapid traverse rate (parameter No.1420)
1: Manual rapid traverse rate (parameter No.1424)

NOTE
This parameter selects a feedrate for reference position return performed without dogs. This parameter also selects a feedrate when manual reference position return is performed according to bit 7 (SJZ) of parameter No.0002 using rapid traverse without deceleration dogs after a reference position is set.

1420	Rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1424	Manual rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rate of manual rapid traverse when the rapid traverse override is 100% for each axis.

NOTE

1. If 0 is set, the rate set in parameter No. 1420 (rapid traverse rate for each axis) is assumed.
2. When manual rapid traverse is selected (bit 0 (RPD) of parameter No. 1401 is set to 1), manual feed is performed at the feedrate set in this parameter, regardless of the setting of bit 4 (JRV) of parameter No. 1402.

5 AUTOMATIC OPERATION

5.1 CYCLE START/FEED HOLD

Overview

- Start of automatic operation (cycle start)

When automatic operation start signal ST is set to 1 then 0 while the CNC is in memory (MEM) mode, DNC operation mode (RMT), or manual data input (MDI) mode, the CNC enters the automatic operation start state then starts operating.

Signal ST, however, is ignored in the following cases:

1. When the mode is other than MEM, RMT, or MDI
2. When the feed hold signal (*SP) is set to 0
3. When the emergency stop signal (*ESP) is set to 0
4. When the external reset signal (ERS) is set to 1
5. When the reset and rewind signal (RRW) is set to 1
6. When MDI RESET key is pressed
7. When the CNC is in the alarm state
8. When the CNC is in the NOT READY state
9. When automatic operation is executing
10. When the program restart signal (SRN) is 1
11. When the CNC is searching for a sequence number.

The CNC enters the feed hold state and stops operation in the following cases during automatic operation:

1. When the feed hold signal (*SP) is set to 0
2. When the mode is changed to manual operation mode (JOG, INC, HND, REF, TJOG, or THND).
3. When an alarm occurs in the CNC (However, in the case that PS, SR, IO, and a part of other alarm occurs, the CNC enters the automatic operation stop state)

The CNC enters the automatic operation stop state and stops operating in the following cases during automatic operation:

1. When a single command block is completed during a single block operation
2. When operation in manual data input (MDI) mode has been completed
3. When a PS, SR, IO alarm and etc. occurs in the CNC
4. When a single command block is completed after the mode is changed to other automatic operation mode or memory edit (EDIT)

The CNC enters the reset state and stops operating in the following cases during automatic operation:

1. When the emergency stop signal (*ESP) is set to 0
2. When the external reset signal (ERS) is set to 1
3. When the reset and rewind signal (RRW) is set to 1
4. When MDI RESET key is pressed

The state of the CNC (automatic operation start, feed hold, automatic operation stop, or reset) is posted to the PMC with status output signals OP, SPL, and STL. See the table in the "Signal" section for details.

- Halt of automatic operation (feed hold)

When the feed hold signal *SP is set to 0 during automatic operation, the CNC enters the feed hold state and stops operation. At the same time, cycle start lamp signal STL is set to 0 and feed hold lamp signal SPL is set to 1. Re-setting signal *SP to 1 in itself will not restart automatic operation. To restart automatic operation, first set signal *SP to 1, then set signal ST to 1 and then to 0.

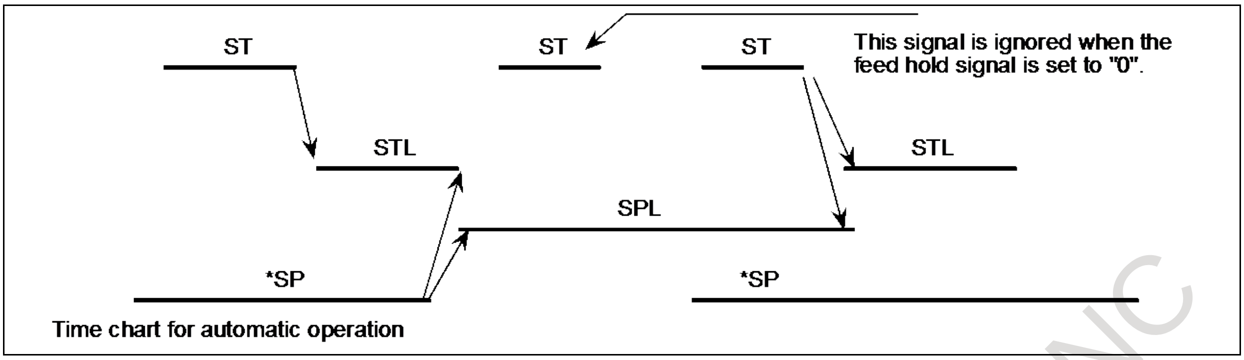


Fig. 5.1 (a)

When signal *SP is set to 0 during the execution of a block containing only the M, S, T, or B function, signal STL is immediately set to 0, signal SPL is set to 1, and the CNC enters the feed hold state. If the FIN signal is subsequently sent from the PMC, the CNC executes processing up until the end of the block that has been halted. Upon the completion of that block, signal SPL is set to 0 (signal STL remains set to 0) and the CNC enters the automatic operation stop state.

(a) During threading
When signal *SP is set to 0 during threading, the CNC enters the feed hold state after executing a non-threading block after the threading blocks.
In lathe system, when signal *SP is set to 0 during threading with the G92 command (threading cycle), signal SPL is immediately set to 1 but operation continues up until the end of the retraction block following threading. When signal *SP is set to 0 during threading with the G32 (M series: G33) command, signal SPL is immediately set to 1 but operation continues until the end of a non-threading block following the threading blocks. (Stopping feeding during threading is dangerous because the amount of cutting will increase.)

(b) During tapping in a canned cycle
When signal *SP is set to 0 during tapping in a canned cycle (G84), signal SPL is immediately set to 1 but operation continues until the tool returns to the initial level or R point level after the completion of tapping.

(c) When a macro instruction is being executed
Operation stops after the currently executing macro instruction has been completed.

Signal

Cycle start signal ST<Gn007.2>

[Classification] Input signal
[Function] Starts automatic operation.
[Operation] When signal ST is set to 1 then 0 in memory (MEM) mode, DNC operation mode (RMT) or manual data input (MDI) mode, the CNC enters the cycle start state and starts operation.
MEM, RMT, or MDI mode

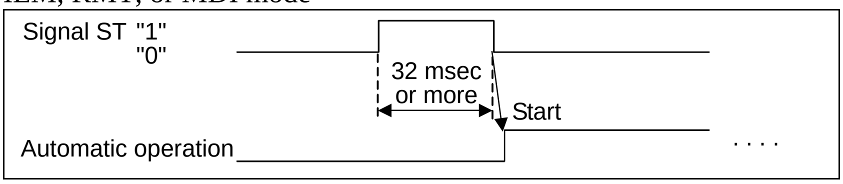


Fig. 5.1 (b)

Feed hold signal *SP<Gn008.5>

[Classification] Input signal
[Function] Halts automatic operation.
[Operation] When signal *SP is set to 0 during automatic operation, the CNC enters the feed hold state and stops operation. Automatic operation cannot be started when signal *SP is set to 0.
MEM, RMT, or MDI mode

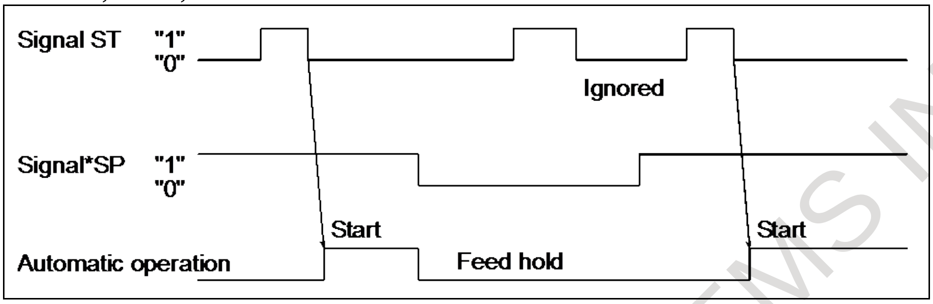


Fig. 5.1 (c)

Automatic operation signal OP<Fn000.7>

[Classification] Output signal
[Function] Notifies the PMC that automatic operation is in progress.
[Output cond.] This signal is set to 1 or 0, according to the state of the CNC, as listed in Table 5.1 (a).

Cycle start lamp signal STL<Fn000.5>

[Classification] Output signal
[Function] Notifies the PMC that automatic operation start is entered.
[Output cond.] This signal is set to 1 or 0, according to the state of the CNC, as listed in Table 5.1 (a).

Feed hold lamp signal SPL<Fn000.4>

[Classification] Output signal
[Function] Notifies the PMC that feed hold state is entered.
[Output cond.] This signal is set to 1 or 0, according to the state of the CNC, as listed in Table 5.1 (a).
Signals OP, STL, and SPL are the signals to inform PMC of the operation condition of CNC.

Table 5.1 (a) Status of operation

Signal name / State of the operation	Cycle start lamp STL	Feed hold lamp SPL	Automatic operation lamp OP
Cycle start state	1	0	1
Feed hold state	0	1	1
Automatic operation stop state	0	0	1
Reset state	0	0	0

(a) Cycle start state
The CNC is executing memory operation or manual data input operation commands.
(b) Feed hold state
The CNC is not executing memory operation nor manual data input operation commands while the commands to be executed remain.
(c) Automatic operation stop state
Memory operation or manual data input operation has been completed and stopped.
(d) Reset state
The automatic operation has been forcibly terminated.

> **NOTE**
> If bit 2 (OPS) of parameter No. 11223 is set to 1, signal OP is set to 1 when a sequence number is searched for by operating keys on the MDI unit in the automatic operation mode (MEM).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn007						ST		
Gn008			*SP					

	#7	#6	#5	#4	#3	#2	#1	#0
Fn000	OP		STL	SPL				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11223						OPS		

[Input type] Parameter input
[Data type] Bit path

#2 OPS In the MEM mode, when a sequence number search operation ([N SEARCH]) is performed, automatic operation signal OP <Fn000.7> is:
0: Kept 0.
1: Set to 1.

Alarm and message

- Self-diagnosis information

During automatic operation, the machine may sometimes show no movement while no alarm is detected. In that case, the CNC may be performing processing or waiting for the occurrence of an event. The state of the CNC can be obtained using the CNC self-diagnosis function (diagnosis data 1000 to 1015).

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Memory operation
	MDI operation
	DNC operation
	Checking by self-diagnosis screen

5.2 RESET AND REWIND

Overview

The CNC is reset and enters the reset state in the following cases:

1. When the emergency stop signal (*ESP) is set to 0
2. When the external reset signal (ERS) is set to 1
3. When the reset and rewind signal (RRW) is set to 1
4. When MDI  key is pressed

When the CNC is reset, the resetting signal (RST) is output to the PMC. The resetting signal (RST) is set to 0 when the resetting signal output time, set with parameter No. 3017, has elapsed after the above conditions have been released.

RST signal output time = Treset (Reset processing time) + (parameter setting value) × 16 msec.

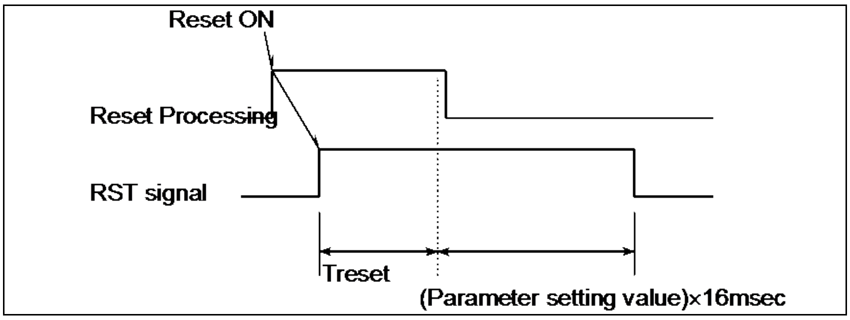


Fig. 5.2 (a)

> ⚠ **CAUTION**
> Treset requires at least 16 msec. This time will be longer on optional configurations.

When the CNC is reset during automatic operation, automatic operation is stopped and tool movement along the controlled axis is decelerated and stopped(CAUTION 1). When the CNC is reset during the execution of the M, S, T, or B function, signal MF, SF, TF, or BF is set to 0 within 100 ms.
Tool movement along the controlled axis is also decelerated and stopped(CAUTION 1) in manual operation (jog feed, manual handle feed, incremental feed, or etc).

> ⚠ **CAUTION**
> When the emergency stop signal (*ESP) is set to 0, the tool is stopped by an emergency stop.

Bit 6 (CLR) of parameter No. 3402 is used to select whether the CNC internal data (such as modal G codes) is cleared or reset when the CNC is reset. Refer to the Appendix F, "Status when turning on power, when cleared, and when reset" in the Operator's Manual for the state of the internal data when cleared or reset.
The following parameters are also used to select how to handle processing for CNC data when the CNC is reset.

- Bit 7 (MCL) of parameter No. 3203
  Whether programs created in MDI mode are erased or stored
- Bit 6 (CCV) of parameter No. 6001
  Whether custom macro variables #100 to #149 (or to #199) are cleared or stored

- Warning indication during a reset in program operation

On the reset state setting for address G, F, H, D, T, S, M, or B (second auxiliary function) when reset occurs, if a reset occurs during block execution in program operation, modal information returns to the state before block execution. Therefore, the warning "MODAL DATA IS CHANGED BY BLOCK STOP" can be displayed so that the operator may notice the modal information is not updated by information of the interrupted block.
To enable or disable the warning, set bit 1 (WMD) of parameter No. 10334.
However, the warning is not displayed if a reset occurs by the reset of the reset & rewind signal RRW.

- Reset & Rewind

When the reset & rewind signal RRW becomes 1, a reset and rewinding are performed.
Rewinding is described below.

1. When automatic operation (DNC operation (RMT)) mode is selected, if a portable tape reader is connected to the selected input/output equipment, the portable tape reader is rewound.
   The rewinding signal RWD is output while the portable tape reader is being rewound. This signal becomes 0 when rewinding is completed.
2. When automatic operation (Memory operation (MEM)) mode or Memory edit (EDIT) mode is selected, the beginning of the selected program is found. When automatic operation (Manual data

input (MDI)) mode is selected, if reset is performed, motion is different according to the state of the program execution and the parameter setting. For details, refer to III-4.2, “MDI OPERATION” in the OPERATOR’S MANUAL (Common to Lathe System / Machining Center System) (B-64694EN). Bit 2 (RWM) of parameter No. 3001 can be used to specify whether the rewinding signal RWD is output.
When RWM is 1, the rewinding signal RWD is kept 1 for 100 ms and then becomes 0. Since the time required for finding the beginning of the program in memory is almost zero, the beginning is already found by the time the rewinding signal RWD becomes 0.

- Reset & Rewind in manual mode

When bit 0 (MRW) of parameter No.10337 is set to 1, the rewind of a program by reset & rewind RRW <Gn008.6> has been possible in manual mode.

Signal

External reset signal ERS<Gn008.7>

[Classification] Input signal
[Function] Reset the CNC.
[Operation] Turning the signal ERS to 1 resets the CNC and enters the reset state. While the CNC is reset, the resetting signal RST turns to 1.

Reset & rewind signal RRW<Gn008.6>

[Classification] Input signal
[Function] CNC is reset and a program under an automatic operation is rewound.
[Operation] As described in the item "Reset & Rewind".

Resetting signal RST<Fn001.1>

[Classification] Output signal
[Function] Notifies the PMC that the CNC is being reset. This signal is used for reset processing on the PMC.
[Output cond.] This signal is set to 1 in the following cases:
1. When the emergency stop signal (*ESP) is set to 0
2. When the external reset signal (ERS) is set to 1
3. When the reset & rewind signal (RRW) is set to 1
4. When MDI RESET key is pressed

This signal is set to 0 in the following case:
- When the resetting signal output time, set with parameter No. 3017, has elapsed after the above conditions have been released and the CNC is reset

Rewinding signal RWD<Fn000.0>

[Classification] Output signal
[Function] Notifies the PMC that the CNC is being rewound.
[Output cond.] As described in the item "Reset and Rewind".

MDI reset confirmation signal MDIRST<Fn006.1>

[Classification] Output signal
[Function] Notifies the PMC that the MDI RESET key has been pressed.
[Output cond.] This signal is set to 1 when:
- The RESET key is pressed when a path for which MDI reset is enabled is selected.

This signal is set to 0 when:

- The MDI RESET key is not pressed.
- A path for which MDI reset is not enabled is selected.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn008	ERS	RRW						

	#7	#6	#5	#4	#3	#2	#1	#0
Fn000								RWD
Fn001							RST	
Fn006							MDIRST	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3001						RWM		

[Input type] Parameter input
[Data type] Bit path

#2 RWM While the beginning of a program in the program memory is being searched for, the rewind signal RWD is:
0: Not output.
1: Output.

3017	Output time of reset signal RST

[Input type] Parameter input
[Data type] Word path
[Unit of data] 16msec
[Valid data range] 0 to 255

When the output time of the reset signal RST is to be extended, set an extended time.
(RST signal output time) =
(Time required for reset processing) + (Parameter setting) × 16 msec

	#7	#6	#5	#4	#3	#2	#1	#0
3203	MCL							

[Input type] Parameter input
[Data type] Bit path

#7 MCL Whether a program prepared in the MDI mode is cleared by reset
0: Not deleted
1: Deleted

	#7	#6	#5	#4	#3	#2	#1	#0
3402		CLR						

[Input type] Parameter input
[Data type] Bit path

#6 CLR Reset button on the MDI unit, external reset signal, reset and rewind signal, and emergency stop signal
0: Cause reset state.
1: Cause clear state.
For the reset and clear states, refer to Appendix in the OPERATOR'S MANUAL.

	#7	#6	#5	#4	#3	#2	#1	#0
6001		CCV						

[Input type] Parameter input
[Data type] Bit path

#6 CCV Common variables #100 to #149(NOTE) cleared by power-off are:
0: Cleared to <null> by reset
1: Not cleared by reset

NOTE
Cleared variables are as the table according to the combination of added options.

		Addition of custom macro common variables (Bit 6 (NCV) of parameter No.8135)	
		Non (NCV=1)	Yes (NCV=0)
Embedded macro	Non	#100 to #149	#100 to #199
	Yes	#100 to #149 and #200 to #499	#100 to #199 and #200 to #499

	#7	#6	#5	#4	#3	#2	#1	#0
8106						CVP		

[Input type] Parameter input
[Data type] Bit

 # 2 CVP By a reset in path other than path 1, the custom macro variables common to paths set to the common variables #100 to #499 cleared by power-off is
0: not cleared to <null>.
1: cleared to <null>. (it depend to the bit 6(CCV) of parameter No.6001.)

NOTE
1 When a reset is made in path 1, it is cleared regardless of the setting of this parameter. (it depend to the parameter CCV)
2 This parameter is invalid for the path that the bit 0(NC1) of parameter No.6020 is 1.
3 This parameter is FANUC initial setting parameter. For details, refer to the appendix "FANUC initial setting / recommendation setting parameter" in this manual.

	#7	#6	#5	#4	#3	#2	#1	#0
10334							WMD	

[Input type] Parameter input
[Data type] Bit

#1 WMD When a reset occurs during program operation, the warning "MODAL DATA IS CHANGED BY BLOCK STOP" is:
0: Not displayed.
1: Displayed.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Status when turning power on, when clear and when reset
	Warning display during a reset in program operation

5.2.1 MDI reset key invalidation function

Overview

It is possible to select whether to make effective the MDI reset key in the automatic operation by parameter settings. As a result, the stop of the automatic operation can be disabled even if the MDI reset key is pressed by mistake in the automatic operation. However, when the automatic operation is not executed or the alarm is generated, the MDI reset key is effective.

Explanation

When the following two conditions are satisfied, this function is effective.
- bit 2 (NRP) of parameter No.13450 is 1.
- Reset key input invalid signal IRTKY<Gn299.7> is 1.

When this function is effective, Table 5.2.1 (a) indicates the relation between the CNC state and the MDI reset key valid (the CNC is reset.) / invalid (the CNC is not reset.).
Even if MDI reset key is pressed, CNC is not reset if MDI reset key is invalid. The program being executed is not interrupted.
However, when the MDI reset key is pushed in the following cases, the CNC is reset regardless of the CNC state shown in Table 5.2.1 (a).
- The alarm is generated.
- The cancel key is pushed at the same time.

Table 5.2.1 (a) The relation between the CNC state and the MDI reset key valid / invalid

Signal name / CNC state	Cycle start lamp STL	Feed hold lamp SPL	Automatic operation OP	MDI reset key valid / invalid
Cycle start state	1	0	1	invalid
Feed hold state	0	1	1	invalid
Automatic operation stop state	0	0	1	invalid / valid (NOTE *)
Reset state	0	0	0	valid

(NOTE *) When bit 3 (NRP) of parameter No.13450 is 1, the MDI reset key is valid in the automatic operation stop state.

Signal

Reset key input invalid signal IRTKY <Gn299.7>

[Classification] Input signal
[Function] The MDI reset key valid / invalid is switched in the automatic operation.
[Operation] 0 : The MDI reset key is valid.
1 : The MDI reset key is invalid.

> **NOTE**
> When bit 2 (NRP) of parameter No.13450 is 1, this signal is effective.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn299	IRTKY							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
13450					NRC	NRP		

[Input type] Parameter input
[Data type] Bit path

#2 NRP When the MDI reset key is pushed in the automatic operation
0: the CNC is reset.
1: the CNC is not reset.

> **NOTE**
> Bit 2 (NRP) of parameter No.13450 and Reset key input invalid signal IRTKY <Gn299.7> must be set “1” for each path to enable this function.

#3 NRC When bit 2 (NRP) of parameter No.13450 is 1, and the MDI reset key is pushed in the automatic operation stop state,
0: the CNC is not reset.
1: the CNC is reset.

When bit 2 (NRP) of parameter No.13450 is 1, the table below indicates the relation between the CNC state and the MDI reset key valid / invalid.

Signal name / CNC state	Cycle start lamp STL	Feed hold lamp SPL	Automatic operation OP	MDI reset key valid / invalid
Cycle start state	1	0	1	invalid
Feed hold state	0	1	1	invalid
Automatic operation stop state	0	0	1	invalid / valid (NOTE *)
Reset state	0	0	0	valid

(NOTE *) When bit 3 (NRP) of parameter No.13450 is 1, the MDI reset key is valid in the automatic operation stop state.

5.3 TESTING A PROGRAM

Overview

Before machining is started, the automatic running check can be executed. It checks whether the created program can operate the machine as desired. This check can be accomplished by running the machine or viewing the position display change without running the machine.

5.3.1 Machine Lock

Overview

The change of the position display can be monitored without moving the machine. When all-axis machine lock signal MLK, or each-axis machine lock signals MLK1 to MLK8 are set to 1, output pulses (move commands) to the servo motors are stopped and the machine coordinates are not updated in manual and automatic operation. The commands are distributed, however, updating the absolute and relative coordinates. The operator can therefore check if the commands are correct by monitoring the position display.

Explanation

- Automatic operation in the machine lock state (M, S, T, and B commands)

Machine lock applies only to move commands along controlled axes. Updating modal G codes or setting a coordinate system is performed normally. M, S, T, and B (2nd auxiliary function) commands are also performed normally.

- Turning on/off the machine lock signal during movement along an axis

When the machine lock signal for an axis is set to 1 during movement along the axis that is not in the machine lock state, the axis is immediately placed in the machine lock state and output pulses (move commands) to the servo motor are stopped. The tool is decelerated and stopped with the automatic acceleration/deceleration function.
On the other hand, when the machine lock signal for an axis is set to 0 during distribution of the move command along the axis in the machine lock state, pulse (move command) output for the axis is immediately restarted. The tool is accelerated with the automatic acceleration/deceleration function.

- Machine lock shift value

When a move command is executed in the machine lock state after once the workpiece coordinate system is fixed, the workpiece coordinate system is shifted from the machine coordinate system because machine coordinates are not updated. This shift amount is called a machine lock shift value. Diagnosis data No.4104 indicates the machine lock shift value.
The machine lock shift value is cancelled by the manual reference position return or the workpiece coordinate system preset operation.

- Machine lock shift value check in automatic operation

If bit 2 (MSC) of parameter No.11501 is 1, when the workpiece coordinate system is shifted by the axis movement in the machine lock state, the alarm PS5559"ILL. AXIS OPERATION (COORD SYS)" is issued. This can prevent forgetting to cancel the shift value by the manual reference position return and the workpiece coordinate system preset operation after the machine lock is released.

- Coordinate position display screen

When a machine lock is being applied to an axis, “L” appears on the left side of the axis address of the axis on the coordinate position display screen.

Limitation

- Outputting the movement state of ax axis

Axis moving signals MV1 to MV8 <Fn102> are useful in the machine lock state.
Axis moving direction signals MVD1 to MVD8 <Fb106> are useless in the machine lock state. These signals maintain their condition before the machine lock.

- Reference position return (G27, G28, and G30)

When the reference position return command (G28), or 2nd to 4th reference position return command (G30), is executed for an axis in the machine lock state, in motion to middle point, distribution and position updating are performed. The tool, however, is not returned to the reference position. The reference position return completion signals (ZP1 to ZP4) are not output.
The reference position return check command (G27) is ignored in the machine lock state.

- Selecting a machine coordinate system (G53)

When a positioning of machine coordinate system (G53) is specified in the machine lock state, move commands are invalid and absolute coordinates and relative coordinates are not updated.
When machine lock is applied during G53 movement, output pulses (move commands) to the servo motors are stopped and machine coordinates are not updated. The absolute and relative coordinates are updated.

- High precision oscillation function

Do not switch machine lock signal to the oscillation axis during oscillation motion, otherwise an alarm PS5050 "ILL-COMMAND IN G81.1 MODE" is issued.

- Manual intervention and return

Do not perform manual intervention in the machine lock state, otherwise the tool might move to wrong position in return operation.

- Tool retract and recover

Do not perform manual retraction in the machine lock state, otherwise the tool might move to wrong position in recovery.

- Program restart / Quick program restart

The tool cannot be returned to a correct position under the following conditions.
- Manual operation is performed in the machine lock state.
- When program restart is specified in the machine lock state, then the machine lock is canceled.

- Type for counting time of tool management function

The count-up operation is not performed in the machine lock state.

- High-speed program check function

During the high-speed program check mode, all-axis machine lock signal MLK and each-axis machine lock signals MLK1 to MLK8 are invalid except for PMC control axis.

- PMC axis control

When bit 0 (MLE) of parameter No.8001 is 1, all-axis machine lock signal MLK and each-axis machine lock signals MLK1 to MLK8 are invalid for PMC-controlled axes. However, each-axis machine lock signals MLK1 to MLK8 become valid by setting bit 1 (MLS) of parameter No.8006 to "1".
All-axis machine lock signal and each-axis machine lock signals are invalid in the following command irrespective of setting parameter MLE and MLS.
- Speed command
- Torque control command

T

- Balance cutting

Balance cutting is not performed in the machine lock state.

- Each-axis machine lock

In some functions (e.g., synchronous control, coordinate system rotation), axes which are not specified move. And relationships between each-axis machine lock signals and effecting axis are various ways.

1 In the following functions, each-axis machine lock signals effect to output pulses (move commands) after conversion.
   - Coordinate system rotation
   - 3-dimentional coordinate conversion
   - Tilted working plane indexing

2 In arbitrary angular axis control, each-axis machine signals are applied to move commands specified in the Cartesian coordinate system (the program coordinate system).
   When each-axis machine lock signal is applied to the perpendicular axis, machine lock is valid to a movement of the perpendicular axis by specifying the perpendicular axis, but invalid to a movement of the perpendicular axis by specifying the angular axis.
   When each-axis machine lock signal is apply to the angular axis, machine lock is valid to the movement of the perpendicular axis and the angular axis by specifying the angular axis.

3 Table 5.3.1 (a) and Table 5.3.1 (b) list about the relational functions to synchronous control.

Table 5.3.1 (a)

Function	Moving axis	MLKx to master axis	MLKx to slave axis
Axis synchronous control	Master axis	Valid	Invalid
	Slave axis	Valid	Invalid
Tandem control	Master axis	Valid	Invalid
	Slave axis	Valid	Invalid
Flexible synchronization control	Master axis	Valid	Invalid
	Slave axis (By specifying master axis)	Valid	Invalid
	Slave axis (By specifying slave axis)	Invalid	Valid
Synchronous control	Master axis	Valid	Invalid
	Slave axis	Invalid	Valid
Superimposed control	Master axis	Valid	Invalid
	Slave axis (By specifying master axis)	Valid	Invalid
	Slave axis (By specifying slave axis)	Invalid	Valid

Table 5.3.1 (b)

Function	Moving axis	MLKx to composite axis 1	MLKx to composite axis 2
Composite control	Composite axis 1	Invalid	Valid
	Composite axis 2	Valid	Invalid

- Functions that cannot be used simultaneously

The following functions are not useful in the machine lock state because they depend on machine coordinates.

- Stored stroke check 1
- Stored stroke check 2, 3
- Stored stroke check before movement
- Position switch
- High-speed position switch

Signal

All-axis machine lock signal MLK<Gn044.1>

[Classification] Input signal

[Function] Places all controlled axes in the machine lock state.

[Operation] When this signal is set to 1, pulses (move commands) are not output to the servo motors for all axes in manual or automatic operation.

Each-axis machine lock signals MLK1 to MLK8<Gn108>

[Classification] Input signal

[Function] Place the corresponding controlled axes in the machine lock state.
These signals are provided for each controlled axis. The signal number corresponds to the number of the controlled axis.

MLK 1

1 . . . Machine lock for the 1st axis
2 . . . Machine lock for the 2nd axis
3 . . . Machine lock for the 3rd axis
:

[Operation] When these signals are set to 1, pulses (move commands) are not output to the servo motors for the corresponding axes (1st to 8th) in manual or automatic operation.

All-axis machine lock check signal MMLK<Fn004.1>

[Classification] Output signal

[Function] Notifies the PMC of the state of the all-axis machine lock signal.

[Output cond.] This signal is set to 1 in the following case:

- When all-axis machine lock signal MLK is set to 1

This signal is set to 0 in the following case:

- When all-axis machine lock signal MLK is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn044							MLK	
Gn108	MLK8	MLK7	MLK6	MLK5	MLK4	MLK3	MLK2	MLK1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn004							M MLK	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11501						MSC		

[Input type] Parameter input

[Data type] Bit path

#2 MSC The machine lock shift value check in automatic operation is:

0: Not applied.

1: Applied.

NOTE
1. This function is invalid for the axis in the machine lock state.
2. The followings can be commanded with the machine lock shift value remained.
   - Automatic reference position return (The middle point is not specified)
   - Workpiece coordinate system preset
   - Local coordinate system setting
   - Machine coordinate system setting
   - Coordinate system setting
   - Workpiece coordinate system setting (Axis movement is not specified)
3. In other automatic operation, if there is no movement in the machine coordinate system, the alarm is not issued.
4. This function is invalid for the dummy axis. (Bit 4(KSV) of parameter No.11802 is set to 1 or bit 0(DMY) of parameter No.2009 is set to 1.)

Alarm and message

Number	Message	Description
PS5559	ILL. AXIS OPERATION (COORD SYS)	When the bit 2(MSC) of parameter No.11501 is set to 1, and the workpiece coordinate system is shifted from the machine coordinate system by the movement command in the machine lock state, the axis movement was commanded. Please operate "reference position return" or "workpiece coordinate system preset" etc.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Machine lock and auxiliary function lock

5.3.2 Dry Run

Overview

Dry run is valid only for automatic operation.
The tool is moved at a constant feedrate regardless of the feedrate specified in the program. This function is used, for example, to check the movement of the tool without a workpiece.

⚠ **CAUTION**
This feedrate depends on the specified parameters, the manual rapid traverse switching signal (RT), manual feedrate override signals (*JV0 to *JV15), and whether the command block specifies rapid traverse or cutting feed, as listed in the table below.

Table 5.3.2 (a)

Manual rapid traverse switching signal (RT)	Program command	
	Rapid traverse	Cutting feed
1	Rapid traverse rate	Dry run feedrate × Jvmax (*2)
0	Dry run speed × JV, or rapid traverse rate (*1)	Dry run feedrate × JV (*2)

Max. cutting feedrate : Setting by parameter No.1430
Rapid traverse rate : Setting by parameter No.1420

Dry run feedrate : Setting by parameter No.1410
JV : Manual feedrate override
Jvmax : Maximum value of manual feedrate override

*1 Dry run feedrate × JV when bit 6 (RDR) of parameter No. 1401 is 1. Rapid traverse rate when parameter RDR is 0.
*2 Clamped by maximum cutting feedrate.

The acceleration/deceleration type and the time constant for this feedrate are listed in the table below.

Table 5.3.2 (b)

Manual rapid traverse switching signal (RT)	Program command	
	Rapid traverse	Cutting feed
1	Rapid traverse type	Cutting feed type
0	Jog feed type, or Rapid traverse type(*3)	Cutting feed type

Time constant for Rapid traverse type : Setting by parameter No.1620
Time constant for Jog feed type : Setting by parameter No.1624
Time constant for Cutting feed type : Setting by parameter No.1622

*3 Jog feed type when bit 6 (RDR) of parameter No. 1401 is 1. Rapid traverse type when parameter RDR is 0.

Signal

Dry run signal DRN<Gn046.7>

[Classification] Input signal
[Function] Enables dry run.
[Operation] When this signal is set to 1, the tool is moved at the feedrate specified for dry run.
When this signal is set to 0, the tool is moved normally.

> ⚠ **CAUTION**
> When the dry run signal is changed from 0 to 1 or 1 to 0 during the movement of the tool, the feedrate of the tool is first decelerated to 0 before being accelerated to the specified feedrate.

Dry run check signal MDRN<Fn002.7>

[Classification] Output signal
[Function] Notifies the PMC of the state of the dry run signal.
[Output cond.] This signal is set to 1 in the following case:
- When dry run signal DRN is set to 1

This signal is set to 0 in the following case:
- When dry run signal DRN is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn046	DRN							

	#7	#6	#5	#4	#3	#2	#1	#0
Fn002	MDRN							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401		RDR	TDR					

[Input type] Parameter input
[Data type] Bit path

#5 TDR Dry run during threading or tapping (tapping cycle G74 or G84, rigid tapping)
0: Enabled
1: Disabled

#6 RDR Dry run for rapid traverse command
0: Disabled
1: Enabled

1410	**Dry run rate**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the dry run rate at the 100% position on the jog feedrate specification dial. The unit of data depends on the increment system of the reference axis. Setting this parameter to 0 results in alarm PS5009, "PARAMETER ZERO (DRY RUN)", being issued.

1420	**Rapid traverse rate for each axis**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Dry run

5.3.3 Single Block

Overview

Single block operation is valid only for automatic operation.
When the single block signal (SBK) is set to 1 during automatic operation, the CNC enters the automatic operation stop state after executing the current block. In subsequent automatic operation, the CNC enters the automatic operation stop state after executing each block in the program. When the single block signal (SBK) is set to 0, normal automatic operation is restored.
Single block operation during the execution of custom macro statements depends on the setting of bits 5 (SBM) and 7 (SBV) of parameter No. 6000, as follows:

SBM=0, SBV=0 : Operation does not stop in the custom macro statements but stops once the next NC command has been executed.

SBM=1 : Operation stops after each block in the custom macro statements.

SBV=1 : Single block operation in custom macro statements is suppressed using macro system variable #3003. Execution of custom macro statements is stopped after each block execution.

> **NOTE**
> 1 In the cutter or tool nose radius compensation mode, the block look-ahead operation is performed even in single-block operation to calculate a path intersection after offsetting. So, a macro statement is also executed when read in advance. Therefore, turn off the cutter or tool nose radius compensation mode, when the single block operation is enabled for custom macro statements by SBM=1 or SBV=1.
> 2 When the single block operation is enabled for custom macro statements by SBM=1 or SBV=1, a custom macro statement block is assumed to be an NC statement block that does not involve movement. Therefore, the motion may differ between the normal operation and the single block operation.

When M99 is executed by the single block operation, operation stops after executing the next block of M99. However, when the single block stop is invalidated by the system variable #3003, and the single block stop is validated again by the system variable #3003, the single block stop is done with the first M99. However, when bit 0 (M99) of parameter No.11648 is set to 1, operation can be stopped with M99.
When the CNC is in the automatic operation stop state during single block operation, the mode can be changed to manual data input (MDI), manual handle feed (HNDL), incremental feed (INC), or jog feed (JOG), by using the mode select signals (MD1, MD2, and MD4).

Signal

Single block signal SBK<Gn046.1>

[Classification] Input signal
[Function] Enables single block operation.
[Operation] When this signal is set to 1, single block operation is performed.
When this signal is set to 0, normal operation is performed.

Single block check signal MSBK<Fn004.3>

[Classification] Output signal
[Function] Notifies the PMC of the state of the single block signal.
[Output cond.] This signal is set to 1 in the following case:
- When single block signal SBK is set to 1

This signal is set to 0 in the following case:
- When single block signal SBK is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn046							SBK	

	#7	#6	#5	#4	#3	#2	#1	#0
Fn004					MSBK			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6000	SBV		SBM					

[Input type] Parameter input
[Data type] Bit path

#5 SBM Custom macro statement

0: Not stop the single block

1: Stops the single block

If you want to disable the single blocks in custom macro statements using system variable #3003, set this parameter to 0. If this parameter is set to 1, the single blocks in custom macro statements cannot be disabled using system variable #3003. To control single blocks in custom macro statements using system variable #3003, use bit 7 (SBV) of parameter No. 6000.

#7 SBV Custom macro statement

0: Not stop the single block

1: Enable/disable single block stop with system variable #3003

		Bit 5 (SBM) of parameter No. 6000	
		0	1
Bit 7 (SBV) of parameter No. 6000	0	Disables single block stop.	Enables single block stop. (With variable #3003, single block stop cannot be enabled/disabled. Single block stop is enabled at all times.)
	1	Enables single block stop. (With variable #3003, single block stop can be enabled/disabled.)	

NOTE

1. In the cutter or tool nose radius compensation mode, the block look-ahead operation is performed even in single-block operation to calculate a path intersection after offsetting. So, a macro statement is also executed when read in advance. Therefore, turn off the cutter or tool nose radius compensation mode, when the single block operation is enabled for custom macro statements by SBM=1 or SBV=1.
2. When the single block operation is enabled for custom macro statements by SBM=1 or SBV=1, a custom macro statement block is assumed to be an NC statement block that does not involve movement. Therefore, the motion may differ between the normal operation and the single block operation.

	#7	#6	#5	#4	#3	#2	#1	#0
11648								M99

[Input type] Parameter input

[Data type] Bit

#0 M99 When the M99 is executed in the single block operation:

0: Operating stops after executing the previous return block.

1: Operating stops before executing the previous return block.

This function is effective only for M99 commanded in the following programs.

- Execution macro of macro executor
- Programs in "//CNC_MEM/SYSTEM" folder (*1)
- Programs in "//CNC_MEM/MTB1" folder (Embedded macro) (*1)
- Programs in "//CNC_MEM/MTB2" folder(*1)
- Programs of O number within the range specified by parameter 11656 and 11658.

(*1) It is effective only to the program put on the right under of each folder. When the subfolder is made and the program is put, parameter M99 is invalid to such a program.

11656	The first O number of the program for parameter M99(No.11648#0) make effective

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 99999999

Specify the first O number for parameter M99(No.11648#0) make effective.
When this parameter is 0, the specification of O number in which parameter M99 is made effective is invalid.

11657	The number of programs for parameter M99(No.11648#0) make effective

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 99999999

Specify the number of programs for parameter M99(No.11648#0) make effective.
When this parameter is 0, the specification of O number in which parameter M99 is made effective is invalid.

CAUTION

⚠ CAUTION

1. Operation in thread cutting
   When the SBK signal turns to 1 during thread cutting, operation stops after execution of the first non-thread cutting block after the thread cutting command.
2. Operation in canned cycle
   When the SBK signal turns to 1 during canned cycle operation, the operation stops at each positioning, approach, drilling and retreat instead of the end of the block. The SPL signal turns to 1 while the STL signal turns to 0, showing that the end of the block has not been reached. When the execution of one block is completed, the STL and SPL signals turn to 0 and the operation is stopped.
3. When a parameter M99(No.11648#0) is set to 1, the tool path might be different because of a single block operation and a continuous operation, when the block where the movements do not exist is commanded just before or just behind M99 in the cutter compensation and tool nose radius compensation mode.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Single block

5.3.4 High-speed Program Check Function

Overview

High-speed program check function has following features.

- The program format check and the stroke limit check are available without axes movements.
- The program check is executed with the maximum feedrate of CNC system and without the acceleration/deceleration regardless of the specified data.
- After the program check is completed, the reference position return is not necessary because the workpiece coordinate system preset is executed automatically at the end of the high-speed program check mode.
- In case that the bit 3 (PGR) of parameter No. 3454 is set to 1, the data, which are changed during the high-speed program check mode, are restored to the data at the start of the high-speed program check mode when ending the high-speed program check mode.

By this, after ending the high-speed program check mode, the automatic operation can be executed with the data before the start of the high-speed program check mode.

Explanation

- Basic operation procedure of the high-speed program check

(1) Set the high-speed program check mode.
(2) Perform a cycle start operation. The high-speed program check is then performed.
(3) After program execution is completed and the check ends, the high-speed program check mode ends.

- Start of High-speed program check mode

The high-speed program check mode is started under both of the following conditions.

- The signal PGCK <Gn290.5> for high-speed program check is 1.
- MEM or DNC mode is selected.

When the mode is except MEM or DNC mode, the high-speed program check mode is not started even if the signal PGCK <Gn290.5> is 1.

- End of High-speed program check mode

The high-speed program check mode is ended under one of the following conditions.

- The signal PGCK <Gn290.5> is changed from 1 to 0.
- A mode except MEM or DNC is selected during the high-speed program check mode.

> **NOTE**
> The alarm PS5365 occurs when the signal PGCK <Gn290.5> is changed while a program is executing.

- Program Check Start

The program check is started when the signal ST <Gn007.2> for cycle start signal is changed from 1 to 0 at the high-speed program check mode.
Then, the program execution is started without axes movements.

- Stroke Limit Check

Following the stroke limit check is executed without axes movements during the high-speed program check mode.

- Stored stroke check 1
- Stored stroke check 2, 3
- Stroke limit check before move

> **NOTE**
> The stroke limit check during the high-speed program check mode is similar to the normal stored stroke limit check. (The alarm 500 etc. occurs at exceeding the stroke limit.)
> As for the details of occurrence alarms, refer to chapter of stored stroke check in Operator's Manual.

- Program Check Feedrate

During the high-speed program check mode, the feedrate of program execution becomes the maximum feedrate of CNC system regardless of the specified feedrate.
The feedrate clamp, override and dry run are not effective.

Table 5.3.4 (a)

Increment system	Program check execution feedrate		
	IS-A	IS-B	IS-C
Millimeter machine (unit: mm/min)	999000	999000	999000
Inch machine (unit: inch/min)	96000	9600	9600
Rotary axis (unit: deg/min)	999000	999000	999000

> ⚠ **CAUTION**
> The program check execution feedrate at circle interpolation is according to the bit 7 (PGF) of parameter No. 1490 setting.

> **NOTE**
> 1 The execution time of dwell is the same as a normal operation.
> 2 The PS011 alarm occurs when there is no F command in a normal operation. However, it is executed at maximum speed without the alarm in the high-speed program check mode even if there is no F command.

- Acceleration / deceleration

During the high-speed program check mode, the acceleration/deceleration after interpolation and before interpolation is not executed regardless of the acceleration/deceleration setting value. (Acceleration/deceleration time constant becomes to 0.)

- Workpiece Coordinate System Preset

When the high-speed program check mode is ended, the workpiece coordinate system preset is executed automatically.
Therefore, the program execution can be done without doing the reference position return after the program check.

> ⚠ **WARNING**
> When the coordinate system is set before the high-speed program check mode is turned on by the work coordinate system setting G92 (machining center system and G code system B or C of the lathe system), G50 (G code system A of the lathe system) and the local coordinate system setting G52, their setting values are deleted by the execution of the work coordinate system preset.

> **NOTE**
> 1 When the bit 3 (PGR) of parameter No. 3454 is set to 0, set the bit 6 (WPS) of parameter No. 3006 to 1 to do workpiece coordinate system preset at the end of the high-speed program check mode. When the bit 3 (PGR) of parameter No. 3454 is set to 1, workpiece coordinate system preset is executed at the end of the high-speed program check mode regardless of the bit 6 (WPS) of parameter No. 3006 setting.
> 2 When the bit 3 (PGR) of parameter No. 3454 is set to 1 in multi-path system, all paths must be in the high-speed program check mode simultaneously. If some paths are in the high-speed program check mode, the automatic operation can not be executed in other paths.

- Machine Coordinate Display For Program Check

During the high-speed program check mode, it is possible to change the display of machine coordinates from the actual machine coordinates to the machine coordinates for the program check by setting bit 7 (PGM) of parameter No.11320.

And it is possible to change the display of machine coordinate for each axis from the actual machine coordinate to the machine coordinate for the program check by setting bit 4 (PGA) of parameter No.3115.

NOTE

1. When the signal EAX* <Gn136> for control axis select (PMC axis control) is set to 1, the machine coordinate of the axis shows the actual machine coordinate regardless of the parameter PGM and PGA setting.
2. The diagnosis data No. 301 (machine position) always shows the actual machine coordinate (the machine position from the reference position) regardless of the parameter PGM and PGA setting.

- Display during High-speed program check mode

During the high-speed program check mode, "PRG-CHK" blinks on the lower right of the CNC display.

- Tool life

If tool life management command or tool management command is specified during the high-speed program check mode, tool life is not counted.

- Auxiliary function

When the bit 6 (PGS) of parameter No. 3001 is set to 0, the auxiliary function is not executed during the high-speed program check mode. This is same as when the signal AFL <Gn005.6> for auxiliary function lock is 1. That is, M, S, T, and B function code signals and strobe signals are not output.
When the bit 6 (PGS) of parameter No. 3001 is set to 1, the M, S, T, and B function code signals and strobe signals are output during the high-speed program check mode. This is same as the normal operation.

⚠ WARNING

Please note that the spindle rotates by S command under the either of following conditions during the high-speed program check mode. To prohibit the rotation of the spindle in the high-speed program check mode, take countermeasure in PMC ladder using the signal PRGMD<Fn290.5>.

- Lathe system
- Constant surface speed control is effective
- T type gear selection method (bit 4 (GTT) of parameter No. 3706 =1)

⚠ CAUTION

1. When the bit 6 (PGS) of parameter No. 3001 is set to 1, the M, S, T, and B codes are output to the PMC. Therefore, if it is not necessary to execute the M, S, T, and B command during high-speed program check mode, please modify ladder sequence not to execute M, S, T, and B code by referring to the signal <Fn290.5> for the high-speed program check mode.
2. If M code (Parameter No.11275), which turns on each axis workpiece coordinate system preset signal, is commanded in the high-speed program check mode with bit 6 (PGS) of parameter No. 3001=0, the alarm PS5364 occurs. In the machine that uses such a program, set bit 6 (PGS) of parameter No. 3001 to 1.
3. In case of using the tool management function, set bit 6 (PGS) of parameter No. 3001 to 1 and execute the window function (code 329) by T code and M code as well as usually operation in order to execute the tool compensation correctly with D99 and H99.

- Data save at start of high-speed program check mode

When the bit 3 (PGR) of parameter No. 3454 is set to 1, the following data are saved at the start of the high-speed program check mode.

- Modal G-code
- F, S, T, M, B, H, D-code
- Tool life management data
- Tool management data
- Variable data changed by programmable data entry (G10)
  - Tool offset values
  - External workpiece origin offset values
  - Workpiece origin offset values (1 to 6)
  - Workpiece origin offset values of additional workpiece coordinate systems (1 to 48 or 1 to 300)
  - Rotary table dynamic fixture offset values (Machining center system)
  - Parameters
  - Tool data of the path interference check function (Lathe system and multi path system)
  - Workpiece coordinate system shift values (Lathe system)
  - Graphic parameters
- Macro variables
  - Local variables #1 to #33
  - Common variables #100 to #149 (#199), #500 to #531 (#999)

NOTE

1. The data of the periodic maintenance screen are not saved.
2. When the bit 6 (PGS) of parameter No. 3001 is set to 1, the M, S, T, and B codes are not saved.
3. In multi path system, the data in all paths is saved when one path becomes the high-speed program check mode.

- Data restore at end of high-speed program check mode

In case that the bit 3 (PGR) of parameter No. 3454 is set to 1, the data, which are changed during the high-speed program check mode, are restored to the data at the start of the high-speed program check mode when ending the high-speed program check mode.
By this, after ending the high-speed program check mode, the automatic operation can be executed with the data before the start of the high-speed program check mode.
The data to be restored are the same as those of described on "Data save at start of high-speed program check mode".

⚠ **WARNING**

1. In multi path system, the data in all paths is saved when one path becomes the high-speed program check mode, and the data in all paths is restored when all paths get out of the high-speed program check mode. That is, when there is different timing of changing the high-speed program check signal PGCK<Gn290.5> in each path, the completion of the high-speed program check mode in a path might influence the operation of other paths. For instance, if the high-speed program check mode of a path is turned off when machining is in progress on another path, it might face dangerous situation because the data of other paths during machining also changes. Therefore, in multi path system, please keep the timing of the state of the high-speed program check mode simultaneously in all paths when you set bit 3 (PGR) of parameter No. 3454 to 1. Please do not have both paths of the high-speed program check mode and paths of normal mode in a multi path system.

> ⚠ **WARNING**
>
> 2 The coordinate value set by the work coordinate system setting G92 (machining center system and G code system B or C of the lathe system), G50 (G code system A of the lathe system) and the local coordinate system setting G52 is not restored.

> ⚠ **CAUTION**
>
> 1 After restoring the data, the absolute coordinate is calculated from machine coordinate by using the restored data.
> 2 In case that the relative position display is preset (the bit 3 (PPD) of parameter No. 3104 is set to 1) in the coordinate system setting, the relative coordinate is not restored and presets same value as the absolute coordinate.
> 3 The data of the periodic maintenance screen are not restored.
> 4 When the bit 6 (PGS) of parameter No. 3001 is set to 1, the M, S, T, and B codes are not restored.
> 5 The save/restore of the data before/after the program check of the high-speed program check function is realized by saving all data before the program check and restoring all data after the program check. Therefore, even when the data is changed without the program command such as the PMC window during the program check, the changed data returns to the value before the program check.

Limitation

- High-speed program check signal

The alarm PS5365 occurs when the signal PGCK <Gn290.5> is changed from 0 to 1 or from 1 to 0 while a program is executing.

- Stroke limit check

When a command that exceeds a stroke limit is specified in the high speed program check mode, an overtravel alarm occurs after the stroke limit is exceeded regardless of the bit 7 (BFA) of parameter No. 1300 setting.
The process of the overtravel alarm cancellation is as follows.
(1) Change to the mode except MEM or DNC.
(2) Push RESET key or input reset signal.

The alarm cancellation of stroke limit check 1 is based on the bit 4 (OF1) of parameter No. 1301. Therefore, when the parameter OF1 is set to 0, the alarm is cancelled by the above process and when this parameter is set to 1, the alarm is cancelled automatically when the high-speed program check mode is ended.

Bit 4 (OF1) of parameter No. 1301:
If the axis is moved into the allowable range after an OT alarm is generated by stored stroke check 1,
0: The alarm is not canceled before a reset is made.
1: The alarm is immediately canceled.

- Stroke limit check before move

When there is a movement command that exceeds the stroke limit in the high-speed program check mode, an alarm of the stroke limit check before move and a usual overtravel alarm might occur at the same time.

- Stored stroke check 2, 3

In case that there is narrow forbidden area, the alarm might not be able to be detected in the block which crosses the forbidden area in the high-speed program check mode.

- Machine coordinate display

As for the axis whose the signal EAX* <Gn136> for control axis select (PMC axis control) is 1, the machine coordinate shows the actual machine coordinate (the machine position from the reference position) regardless of the bit 7 (PGM) of parameter No. 11320 and bit 4 (PGA) of parameter No. 3115 setting.

- Workpiece coordinate system preset

Please set the bit 6 (WPS) of parameter No. 3006 to 1 to do workpiece coordinate system preset automatically at the end of the high-speed program check mode.
When the workpiece coordinate system preset is commanded by the soft key operation or input signal during the high-speed program check mode, the workpiece coordinate system is established based on the actual machine coordinate.

> ⚠ **WARNING**
> When the coordinate system is set before the high-speed program check mode is turned on by the work coordinate system setting G92 (machining center system and G code system B or C of the lathe system), G50 (G code system A of the lathe system) and the local coordinate system setting G52, their setting values are deleted by the execution of the work coordinate system preset.

- Reference Position Return (G28)

The alarm PS5364 occurs when the reference position is not established and G28 is specified during the high-speed program check mode.

- Machine Lock Signal

During the high-speed program check mode, all-axis machine lock signal MLK <Gn044.1> and each-axis machine lock signals MLK1 to MLK8 <Gn108> are invalid except for PMC control axis.

- Control axis select signal (PMC axis control)

In case that the bit 3 (PGR) of parameter No. 3454 is set to 1, the alarm PS5364 occurs when the signal EAX* <Gn136> for control axis select (PMC axis control) is changed from 0 to 1 or from 1 to 0 during the high-speed program check mode.

- Save / restore of data

The save/restore of the data before/after the program check of the high-speed program check function is realized by saving all data before the program check and restoring all data after the program check.
Therefore, even when the data is changed without the program command such as the PMC window during the program check, the changed data returns to the value before the program check.

- Restore of data

When the power off request alarm is generated during the high-speed program check mode and the CNC power is turned off immediately, the changed data still remain. Therefore, if a restore of data is necessary, turn off the CNC power after ending the high-speed program check mode and restoring the saved data.

- Programmable parameter input (G10)

The bit 3 (PGR) of parameter No. 3454 and the bit 6 (PGS) of parameter No. 3001 can not be changed by programmable parameter input (G10). When G10 L52 is commanded to these parameter bits, the alarm PS5364 occurs.

- Cycle start

In case that the bit 3 (PGR) of parameter No. 3454 is set to 1, the signal ST <Gn007.2> for cycle start are invalid during the data saving and restoring.

- Unusable functions

The high-speed program check function can not be used in combination with the following functions.

- Angular axis control
- Flexible synchronization control

If some of above functions are effective, the alarm PS5364 occurs at the start of the high-speed program check mode.

- Unavailable command

Following functions can not be specified during the high-speed program check mode. If the following functions are specified, the alarm PS5364 occurs.

- Electronic gear box

- Restriction of commands

During the high-speed program check mode, the axes movements of the following functions can not be checked with this function.

- PMC axis control

  The axis whose the signal EAX* <Gn136> for control axis select (PMC axis control) is 1 is not checked in the high speed program check mode. The stroke limit check of the PMC axis is done by the actual machine coordinate.

- Auxiliary function

When the bit 6 (PGS) of parameter No. 3001 is set to 1, the M, S, T, and B codes are not saved/restored at the start/end of the high-speed program check mode.
Therefore, the M, S, T, and B codes, which are output during the high-speed program check mode, are effective after the end of the high-speed program check mode.
When the bit 6 (PGS) of parameter No. 3001 is set to 1, the M, S, T, and B codes are output to the PMC.
Therefore, if it is not necessary to execute the M, S, T, and B command during high-speed program check mode, please modify ladder sequence not to execute M, S, T, and B code by referring to the signal <Fn290.5> for the high-speed program check mode.

- Tool management function

When instructing in D99 and H99 by the tool management function, the window function notifies CNC side of the tool at the spindle position and the tool compensation is executed by using the tool offset amount of the tool. Therefore, in order to execute the tool compensation correctly with D99 and H99 in the tool management function, please set bit 6 (PGS) of parameter No. 3001 to 1 and execute the window function (code 329) by T code and M code in PMC ladder as well as usually operation.

- Multi path system

In case that save/restore of data is effective (bit 1 (PGR) of parameter No. 3454 is set to 1)
please make the state of the high-speed program check mode the same in all paths in the multi path system.

- Synchronous / Composite control

In case that you execute the high-speed program check for program that uses the synchronous/composite control, please set paths related to synchronous/composite control into high-speed program check mode. When all of them are not under high-speed program check mode, the cycle start is disregarded and the program cannot be started.

- Superimposed control

> ⚠ **WARNING**
> When the Superimposed control is executed in the high-speed program check mode, the stroke limit is not checked correctly.

Signal

High-speed program check signal PGCK<Gn290.5>

[Classification] Input signal

[Function] Specify the high-speed program check mode.

[Operation] When this signal is 1, the high-speed program check mode is effective.
When this signal is 0, the high-speed program check mode is not effective.

> **NOTE**
> The alarm PS5365 occurs when the signal PGCK <Gn290.5> is changed while a program is executing.

High-speed program check mode signal PRGMD<Fn290.5>

[Classification] Output signal

[Function] Notify that the high-speed program check mode is active.

[Operation] This signal turns to 1 under both of the following conditions.

- The signal PGCK <Gn290.5> is 1.
- MEM or DNC mode is selected.

In case that the bit 3 (PGR) of parameter No. 3454 is set to 1, the data are saved under both of the above conditions. This signal turns to 1 when the save of data is completed.
This signal turns to 0 under one of the following conditions.

- The signal PGCK <Gn290.5> is 0.
- A mode except MEM or DNC is selected.

In case that the bit 3 (PGR) of parameter No. 3454 is set to 1, the data are restored under one of the above conditions. This signal turns to 0 when the restore of data is completed.

High-speed program check saving data signal PCKSV<Fn290.4>

[Classification] Output signal

[Function] Notify that the data which are changed by automatic operation during the high-speed program check mode are saved.

[Operation] This signal turns to 1 under the following conditions.

- The save of the data, which are changed by automatic operation, is complete at the start of the high-speed program check mode.

This signal turns to 0 under the following conditions.

- The restore of the data, which are changed by automatic operation, is complete at the end of the high-speed program check mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn290			PGCK					

	#7	#6	#5	#4	#3	#2	#1	#0
Fn290			PRGMD	PCKSV				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1490	PGF							

[Input type] Parameter input

[Data type] Bit path

#7 PGF The feedrate specified for circular interpolation in the high-speed program check mode is:

0: The dry run feedrate.
At this time, manual feedrate override signals *JV0 to *JV15 <Gn010 to Gn011> can be used.

1: The maximum feedrate specified by the CNC.

> ⚠ **CAUTION**
> If this parameter is set to 1, feedrate clamp, override, and dry run for circular interpolation is disabled. If a movement around a stroke limit is specified, therefore, a stroke limit check cannot sometimes be made correctly.

	#7	#6	#5	#4	#3	#2	#1	#0
3001		PGS						

[Input type] Parameter input
[Data type] Bit path

#6 PGS In the high speed program check mode, M, S, T, and B codes are:
0: Not output.
1: Output.

> ⚠ **CAUTION**
> 1 If this parameter is set to 1, M, S, T, and B codes are neither saved nor restored at the start and end of the high speed program check mode.
> Accordingly, M, S, T, and B codes output in the high speed program check mode remain valid even after the high speed program check mode ends.
> 2 If this parameter is set to 1, M, S, T, and B codes are output to the PMC in the high speed program check mode.
> Therefore, when M, S, T, and B commands need not be executed in the high speed program check mode, create a ladder sequence that references the high speed program check mode signal <Fn290.5> and suppresses the execution of any of the M, S, T, and B codes.
> 3 If this parameter is set to 1, the operation of the M, S, T, and B codes depends on the status of auxiliary function lock signal AFL <Gn005.6>.
> 4 In the high speed program check mode, an attempt to rewrite this bit parameter by using G10 results in an alarm PS5364, “ILLEGAL COMMAND IN PROGRAM CHECK”.

	#7	#6	#5	#4	#3	#2	#1	#0
3006		WPS						

[Input type] Parameter input
[Data type] Bit

#6 WPS Each axis workpiece coordinate system preset signal:
0: Disabled.
1: Enabled.
When this parameter is set to 1, a workpiece coordinate system is preset after the end of the high speed program check mode.

	#7	#6	#5	#4	#3	#2	#1	#0
3454					**PGR**			

[Input type] Parameter input
[Data type] Bit path

#3 PGR In the high speed program check mode, data modified during automatic operation is:
0: Neither stored nor restored.
1: Stored and restored.

If this parameter is set to 1, when the high speed program check mode ends, data modified during automatic operation in the high speed program check mode can be restored to the state present before the start of the high speed program check mode.
After the high speed program check mode ends, therefore, it is possible to perform automatic operation in the state present before the start of the high speed program check mode.

> ⚠ **WARNING**
> In a multi-path system, when one of the paths enters the high-speed program check mode, data of all paths is stored. After there is no path left in the high-speed program check mode, the data of all paths is restored at a time. This means that when the time to change the status of the high-speed program check input signal PGCK<Gn290.5> differs among the paths, the end of the high-speed program check mode in a path may affect the operation of other paths. For example, if the high-speed program check mode of a path is turned off when machining is in progress on another path, data of the path on which machining is in progress is also restored, which poses serious danger. Therefore, when bit 3 (PGR) of parameter No. 3454 is set to 1 in a multi-path system, the status of the high-speed program check mode must be made consistent throughout the paths. Make sure that paths placed in the high-speed program check mode and paths placed in the normal mode are not present at the same time.

> ⚠ **CAUTION**
> 1 When this parameter is set to 1, the workpiece coordinate system preset is executed at the end of high-speed program check mode regardless of the setting of the bit 6 (WPS) of parameter No. 3006.
> 2 The absolute coordinate and relative coordinate of the axis, whose the signal EAX* <Gn136> for control axis select (PMC axis control) is 1 at the start/end of high-speed program check mode are not saved/restored.
> 3 The data of the periodic maintenance screen are not saved/restored.
> 4 During the high-speed program check mode, if G10 command is specified to this parameter bit, the alarm PS5364 occurs.
> 5 During the high-speed program check mode, if the EAX* <Gn136> is changed from 0 to 1 or from 1 to 0, the alarm PS5364 occurs.

	#7	#6	#5	#4	#3	#2	#1	#0
11320	PGM							

[Input type] Parameter input
[Data type] Bit path

#7 PGM In the high speed program check mode, the machine position is displayed with:
0: Actual machine coordinates. (Machine position relative to the reference position)
1: Machine coordinates for the program check.

	#7	#6	#5	#4	#3	#2	#1	#0
3115				PGAx				

[Input type] Parameter input
[Data type] Bit axis

#4 PGAx In the high speed program check mode, the machine position on each axis is:
0: Displayed according to the setting of bit 7 (PGM) of parameter No. 11320.
1: Displayed with machine coordinates used for program checking.

NOTE
1 When PMC axis select signal EAX*<Gn136> is set to 1 for an axis, the actual machine coordinate value on the axis is indicated regardless of the settings of bit 7 (PGM) of parameter No. 11320 and bit 4 (PGA) of parameter No. 3115.
2 In diagnosis data No. 301 (machine position), actual machine coordinates are always displayed regardless of the settings of bit 7 (PGM) of parameter No. 11320 and bit 4 (PGA) of parameter No. 3115.

Alarm and message

Number	Message	Description
PS5364	ILLEGAL COMMAND IN PROGRAM CHECK	(1) An invalid G code is specified in high-speed program check mode. (2) Angular axis control is available. (3) Following operation is done. - During the high-speed program check mode, an axis, which is never established the reference position, is returned to the reference position. (4) During the high-speed program check mode, the signal EAX* <Gn136> is changed. (5) During the high-speed program check mode, G10 is commanded to the bit 3 (PGR) of parameter No. 3454. (6) During the high-speed program check mode, G10 is commanded to the bit 6 (PGS) of parameter No. 3001.
PS5365	NOT CHANGE OF PROGRAM CHECK MODE	Switching of high-speed program check input signal PGCK<Gn290.5> was performed during execution of the program.

5.3.5 Manual Handle Retrace

Overview

In this function, the program can be executed both forward and backward with a manual handle (manual pulse generator) under automatic operation.

Therefore, errors of a program, interference, and so on can be checked easily by working a machine actually.

- Checking mode

In this mode, the program can be executed forward and backward and the program can be checked.
To change to the checking mode, it is necessary to change the mode to the MEM mode, and the checking mode signal MMOD<Gn067.2> is set to 1. This function makes the data to execute the program backward when the program is executed forward in the checking mode.
To work a machine synchronizing with a pulse generated by a manual handle in the checking mode, the manual handle check signal MCHK <Gn067.3> is set to 1 in addition to the above-mentioned. As a result, it becomes possible to check the program with a manual handle.

> **NOTE**
> During the checking mode, it is not possible to change the parameter and offset.

- Forward movement with a manual handle

The "forward movement" is that the program is executed forward by turning a manual handle in the positive direction (when the manual handle check signal is set to 1.) or in no relation to rotation of a manual handle (when the manual handle check signal is set to 0.).
When the manual handle check signal is set to 1, the execution speed of the program is proportional to the number of rotations of a manual handle. The program is executed forward rapidly when a manual handle is turned to the positive direction rapidly. And, the program is executed forward slowly when a manual handle is turned to the positive direction slowly. The distance magnification traveled per pulse from manual handle can be switched as same as a usual manual handle feed function.
When the manual handle check signal is set to 0, the execution of the program is controlled as same as an automatic operation.

- Backward movement

The "backward movement" is that the program executed forward once is executed backward by turning a manual handle in the negative direction.
The program can be executed backward only for the block executed forward. And, the number of blocks for it is about 190 blocks. This block number changes by the content of the commanded program.
The program is executed backward rapidly when a manual handle is turned to the negative direction rapidly. And the program is executed backward slowly when a manual handle is turned to the negative direction slowly. The distance traveled per pulse from manual handle can switch magnification as well as a usual manual handle feed.

> **NOTE**
> In the system which exceeds 4 paths and 20 axes, the maximum number of blocks which can move backward becomes about 97 blocks.

Explanation

- Control by the manual handle

Program execution start

The checking mode signal MMOD<Gn067.2> is set to 1 in the MEM mode in order to change the checking mode. Then, the program execution is begun by turning ST signal from 1 to 0.
If the manual handle check signal MCHK <Gn067.3> is set to 1 at this time, the execution of the program is controlled by a manual handle. The program is executed synchronizing with rotation of a manual handle.
When a manual handle check signal MCHK<Gn067.3> is set to 0, it is controlled as usual execution.
When check mode signal MMOD<Gn067.2> is set to 1 during the operation of the program, it is enabled a check mode from the block that next buffering is done.
That is, even if check mode signal set to 1, check mode is not always enabled at once.

When check mode is enabled, check mode confirmation signal MMMOD<Fn091.3> is set to 1.

> **NOTE**
> After the signal MMOD is turned to 0 during the execution of the program, the program cannot be executed forward and backward.

Control with the manual handle

The value of the parameter No.6410 and the scale factors decide the moving speed of the machine by one pulse generated by a manual handle.
When a manual handle is turned, the actual movement speed of the machine is as follows.
[Feedrate command value] × [Number of the handle pulse per a second]× [Handle magnification] × ([Parameter setting value]/100) × (8/1000) (mm/min or inch/min)
Example)

When feedrate command value is 30mm/min, handle magnification is 100, parameter No.6410 is set to 1 and manual pulse generator is rotated at 100 pulse/rev, the feedrate of axis is decided as follow.
[Feedrate]=30[mm/min] × 100[pulse/s] × 100 × (1/100) × (8/1000)[s]=24[mm/min]

When the feedrate exceeds the override 100% feedrate by turning a manual handle rapidly, the feedrate is clamped at the speed of override 100%. That is, if the pulse of the following formula exceeds 1, the feedrate is clamped.

[Number of the handle pulse per a second]× [Handle magnification] ×
([Parameter setting value]/100) × (8/1000)

The rapid traverse feedrate is clamped at 10%. However, the feedrate of the rapid traverse is clamped at 100% when the bit 0 (RPO) of parameter No. 6400 is set to 1.
And if parameter No.6405 is set to an optional value, it can be clamped to override by nearly optional value.
When the parameter No.6405 is set to larger value than "100", it is clamped to nearly 100%.
When parameter No.6405 is set to 0, the setting of bit 0 (RPO) of parameter No. 6400 becomes valid.
The single block signal and the feed hold signal are effective in the checking mode. When the execution of a program is stopped by the single block stop or the feed hold stop, it is necessary to turn ST signal from 1 to 0 in order to restart the program.
In the block with the movement and the block of dwell, the execution speed of the program can be controlled by turning a manual handle. As for the block of neither movement nor dwell such as the block of only address M, S, T, and F, the program advances to the following block even if a manual handle does not turn.
The rotation of the spindle does not synchronize with a pulse of a manual handle. During the checking mode, the spindle rotates at the specified rotation speed. As for the feed per revolution, a program is executed at the feedrate which was converted from the rotation speed of the spindle to the corresponding feed per minute inside CNC.

> **NOTE**
> The manual handle used by this function is always the first. The 2nd and 3rd manual handle cannot be used in this function.

Forward movement and backward movement with a manual handle

The program is executed forward when a manual handle is turned to the positive direction. And, the program is executed backward when a manual handle is turned to the negative direction.
The program is executed backward if a manual handle is turned to the negative direction in executing the program forward.
When a manual handle keeps being turned in a negative direction, the program is executed backward and the execution stops in the block of O number. Then, if a manual handle is turned to the positive direction, the program is executed forward again.
Even if a manual handle controls the program execution, the program is executed forward in no relation to a pulse generated by a manual handle on setting the manual handle check signal to 0.

The rotation amount of the manual handle is not accepted as an amount of the movement while changing forward movement and backward movement.

Program end

When the block of M2 or M30 is executed, the manual handle retrace ends. Execute backward movement from the block of M2 or M30 is not possible.
When the execution of the program ends, the checking mode signal and the manual handle check signal must be set to 0.
In 2 path control system, FIN signal must not be set to 1 when the block of M2 or M30 is executed in only one of paths. After the block of M2 or M30 has been executed in both paths, FIN signal is set to 1. (Except for the block of waiting M code is commanded before M2 or M30 in both paths.)

- Notice of the operation

- Dry-run can not operate during the checking mode. Dry-run signal must be set to 0.
- Automatic operation starts immediately with the feedrate commanded by the program, when the checking mode signal or the synchronous operation with handle signal is turned off during executing the program in the checking mode.
- While a program is executing, the edit of the program and the change of the parameter and the offset must not be done.
- While the program is being executed backward or blocks once executed backward are being executed forward again, do not change the checking mode signal MMOD <Gn067.2>.
- The single block operation is also possible. However, even if the manual handle is operated in single block stop state, neither forward movement nor backward movement is done. If cycle start is done, the continuation of the program is executed in the same direction before a single block stop, and the manual handle becomes effective.

- Backward movement of each code

All modal information of G, T, and S-code is memorized in executing the program forward. And, the memorized data of the modal G, T, and S-code are used in executing the program backward.
As for M-codes, they are grouped and the modal information is managed by parameter No.6411 to 6490. Therefore, M-code can be executed backward according to the information. As for the modal information of the M-code, a change in each group is memorized in the execution data.
As for the codes except for G, M, S, and T, the same code is output between forward movement and backward movement.

- G-code

If G-code that changes modal information is commanded in backward movement, the modal information of previous block is executed.
Example)

```
N1G99;
N2G01X_F_;
N3X_Z_;
N4G98;    .................... backward movement starts from this block
N5X_Y_Z_;
```

If backward movement starts from N4 block, the modal information is changed from G98 to G99 and G99 is executed from N3.
G-code with a movement is traced along the route opposite to forward movement.
G-code that can be command in executing the program backward is as follows.
The other G-codes cannot be command in executing the program backward.
The G-codes in the G-code system B and C also can be used.

Lathe system

G00 G01 G02 G03 G04 G10 G11
G22 G23 G25 G26 G27 G28 G30
G40 G41 G42 G50 G53 G61 G63
G64 G65 G68 G70 G71 G72 G73
G75 G80 G83 G85 G87 G89 G90
G94 G96 G97 G98 G99
(G-code system A)

Machining center system

G00 G01 G02 G03 G04 G10 G11
G22 G23 G25 G26 G27 G28 G30
G40 G41 G42 G43 G44 G49 G53
G61 G63 G64 G65 G73 G76 G80
G81 G82 G83 G85 G86 G87 G88
G89 G92 G94 G95 G96 G97

NOTE

1. In Small-Hole Pecking Drilling Cycle(G83), backward movement is prohibited.
2. In forward movement of Boring Cycle(G88), the sequence of actions at bottom of hole is shown as follows (dwell → stop of spindle motor → hold state). But in backward movement, that is (rotation of spindle → hold state → dwell after restart).
3. G68 for the lathe system means mirror image for double turret.

- M-code

If there is M-code in the same group is commanded in previous blocks, modal information of the M-code, commanded at the last in previous blocks, is output.
If no M-code is commanded in previous blocks, the M-code set to the first parameter in the same M-code group is output.
If M-code is not set to group M-code in parameter, the same M-code is output in backward movement.
If the bit 5 (RVN) of parameter No. 6400 is set to 1, the backward movement is prohibited when the M-code, which is not set to group M-code, is commanded in backward movement.

NOTE

When setting the bit 5 (RVN) of parameter No. 6400, backward movement prohibition is enabled except the M-code which was set in the grouping but backward movement can be enabled for the following M-code exceptionally.

1. Subprogram Call by M98/M99.
2. Subprogram Call using an M code
3. Macro Call using an M code
4. Waiting M code
5. M0

Example)

Output of M-codes that are set to groups by parameters in backward movement
Setting of parameters:
No.6400#2=1, #3=0 (5 M-codes/group and 16 groups)

No.6411=100
No.6412=101
No.6413=102 Group A
No.6414=103
No.6415=104
No.6416=200
No.6417=201
No.6418=202 Group B
No.6419=203
No.6420=204

Program O10 is executed in forward movement from N1 to N15 and backward movement is executed from N15. In backward movement, the output of M-codes is show as next table.

	Forward movement	**Backward movement**
O10;		
N1G4X1.;		
N2M101;	M101	M100 (*1)
N3G4X1.;		
N4M204;	M204	M200 (*1)
N5G4X1.;		
N6M104;	M104	M101 (*2)
N7G4X1.;		
N8M300;	M300	M300 (*3)
N9G4X1.;		
N10M200;	M200	M204 (*2)
N11G4X1.;		
N12M0;	M0	M0 (*3)
N13G4X1.;		
N14M102;	M102	M104 (*2)
N15G4X1.;	Backward movement starts from this block	
M2;		

*1 No M-code in the same group is commanded before this block, so the M-code, which is set in the 1st. parameter of the same group, is output.
*2 M-code in the same group is commanded before this block, so the M-code, which is commanded at the last before this block, is output.
*3 M-code is not set to group M-code, so the same M-code is output.

- S and T-code

A modal value of the previous block is output.
When movement command and S-code or T-code is commanded in the same block, the timing of the output of the S-code and T-code is different. Because, the timing where S-code and T-code are output at the forward movement is different from that at the backward movement. By setting the bit 7 (STO) of parameter No. 6401 to 1, the timing of the output of S and T code at the forward movement is the same as the one at the backward movement.

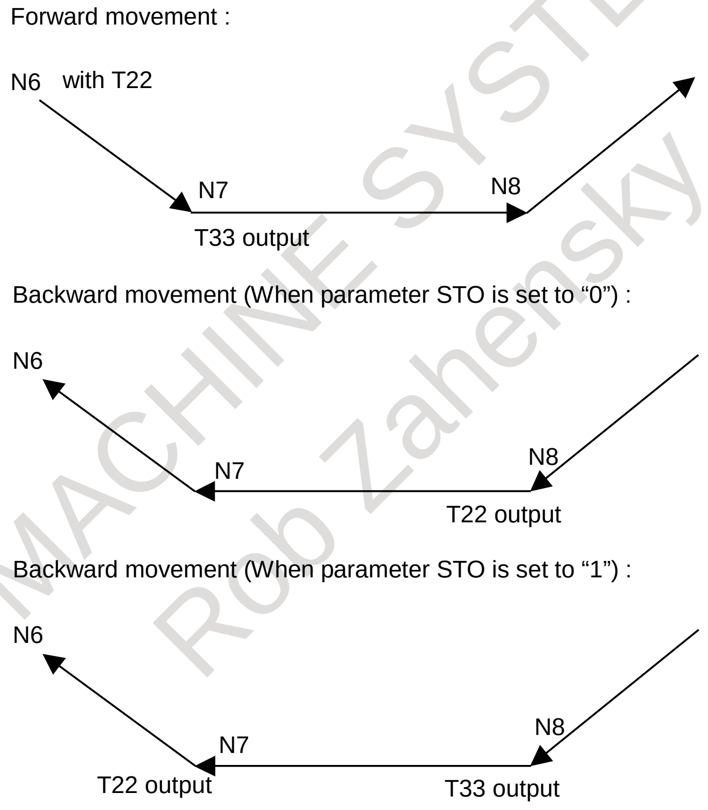
Example)

T-code output timing at the backward movement
T-code is output as follows when the program proceeds backward after the forward movement to N8 block.

	Forward movement	Backward movement	
		Parameter STO=0	Parameter STO=1
O1000;			
N1G98G00X0Z0;			Default T output
N2G00X-10.T11;	T11 output	Default T output	
N3G00X100.;			T11 output
N4G00X10.Z20.T22;	T22 output	T11 output	No T code output
N5G00X30.Z30.;			
N6G00X-10.Z-20.;			T22 output
N7G00X50.Z40.T33;	T33 output	T22 output	T33 output
N8G04X5.;		(Backward start)	(Backward start)
M30;			

The “Default T” means a T-code status at N1 block in forward movement. If the status is T0, “T0” signal is output as “Default T” in the backward movement.

The timing of T-code output of N7 and N8 in O1000 shown in the example above is as follows.



- Direction change prohibition

The direction change prohibition is a state not changing the direction where the program is executed.
In the state, even if the rotating direction of a manual handle is reversed, the reversed rotation is ignored.
A manual handle must be rotate in the same direction as present direction for removing this state.
The direction change prohibition can be confirmed by output signal MNCHG<Fn091.1>.

It becomes the change prohibition state under the following condition.

- While the block with the code waiting for FIN is executing
- After a block has done and until the next block begins to operate
- During thread cutting
- Modal G code of G68.1, G68 and G51.2

- The block with the axis that ends movement earlier in the block with G02 or nonlinear type position etc.
- During a time wait due to block switching (in a multi-path system only. See "Time wait in a multi-path system".)

- Backward movement prohibition

The backward movement prohibition is a state that the program cannot be executed from a certain block backward. In this state, the negative rotation of a manual handle is ignored, and the only positive rotation is effective. The program must be executed forward by rotating a manual handle in the positive direction for removing this state.
The backward movement prohibition can be confirmed by output signal MRVSP<Fn091.2>.
If the following blocks are executed in backward movement, backward movement is prohibited.

- Program number block of main program (except subprogram and macro program)
- Over the maximum number of the blocks for reverse movement
- The block including backward movement prohibition G-code (which is not described in the paragraph "G-code")
- The block which is executed while in modal including backward movement prohibition G-code (which is not described in the paragraph "G-code")

- Status display

In manual handle retrace, the status of manual handle retrace is displayed on clock display of CNC state display line. This status display is displayed during the execution of manual handle retrace. The clock is displayed usually.

When the all condition is filled, "M.H.RTR" is displayed on clock display of CNC state display line. This status is displayed by the color of color number 3 (Title, Axis name and O/N No. are the same color). The screen display is as shown in Fig. 5.3.5 (a). When the following conditions are not full, the clock is displayed.

- When bit 2 (CHS) of parameter No. 6401 is set to 0:
  1) Status display disable/enable bit 6 (HST) of parameter No. 6401 is set to 1.
  2) Check mode confirmation signal MMMOD<Fn091.3> is set to 1.

- When bit 2 (CHS) of parameter No. 6401 is set to 1:
  1) Status display disable/enable bit 6 (HST) of parameter No. 6401 is set to 1.
  2) Cycle start signal STL<Fn000.5> is set to 1.
  3) Check mode signal MMOD<Gn067.2> is set to 1.
  4) Manual handle check signal MCHK<Gn067.3> is set to 1.

Fig. 5.3.5 (a) "M.H.RTR." status display

Besides, when reverse movement prohibition signal MRVSP<Fn091.2> is set to 1, the "NO RVRS." is displayed. This status is displayed by blinking/reversing in the color of color number 1 (ALARM is the same color). The screen display is as shown in Fig. 5.3.5 (b). When reverse movement prohibition signal MRVSP<Fn091.2> is set to 0, the “M.H.RTR " is displayed again.

Fig. 5.3.5 (b) “NO RVRS." status display

Besides, when direction change prohibition signal MNCHG<F0091.1> is set to 1 and the direction of program’s execution is changed by manual handle, this status display changes from "M.H.RTR" to "NO.CHAG".

This status is displayed by blinking/reversing in the color of color number 3 (INPUT KEY, O/N NO. and STATUS are the same color). The screen display is shown as Fig. 5.3.5 (c). When the program is executed in the direction as the same as before by manual handle or direction change prohibition signal MNCHG<Fn091.1> is set to 0, the " M.H.RTR " is displayed again.

Moreover, when bit 1 (FWD) of parameter No. 6400 is set to 1 and the program is executed to change direction by manual handle, this status display changes from "M.H.RTR" to "NO.CHAG".

NOTE

When the improvement of direction change movement in auxiliary function output block is enabled, the state of direction change prohibition signal MNCHG<Fn091.1> is set to 1 and direction change is possible. Therefore please note that the status display of the direction change prohibition is displayed though it is possible to direction change.

Fig. 5.3.5 (c) "NO.CHAG." status display

Time constant for acceleration/deceleration after interpolation

The time constant for acceleration/deceleration after interpolation of the manual handle retrace mode (parameter No. 6495, No. 6496 or No. 6497) can be set.

When check mode confirmation signal MMMOD<Fn091.3> and handle available signal in checking mode MCHK<Gn067.3> are set to "1", and a manual handle is operated, these parameters are enabled.

NOTE

1. When effective and invalidity of handle of the manual handle retrace mode change, the distribution of the pulse is stopped once. Afterwards, a time constant for acceleration/deceleration after interpolation changes, and the distribution of the pulse is restarted. The change of enabled and disabled of handle of the manual handle retrace mode is disregarded between from the distribution stops of the pulse to a time constant for acceleration/deceleration after interpolation change. When a value is set even by one axis of parameters (No.6495-No.6497), it is sure to become the above-mentioned operation.
2. A time constant for acceleration/deceleration after interpolation for rigid tapping (parameter (No.5261-No.5264 and No.5271-No.5274)) is enabled in rigid tapping. A time constant for acceleration/deceleration after interpolation for this function is disabled.
3. A time constant for acceleration/deceleration after interpolation for the skip function for each axis is disabled when a time constant for acceleration/deceleration after interpolation for this function is enabled.

> **NOTE**
> 4 A linear acceleration/deceleration time constant(Parameter No.8192) in rapid traverse for each of the axes (master and slave axes) under superimposed control is disabled when a time constant for acceleration/deceleration after interpolation for this function is enabled.
> 5 A time constant for acceleration/deceleration after interpolation for positioning by optimum accelerations is disabled when a time constant for acceleration/deceleration after interpolation for this function is enabled.

Limitation

- Movement in automatic operation by DNC operation mode(RMT)

In the automatic operation by DNC operation mode(RMT), the backward movement is prohibited though the forward movement is enable.

- Movement in subprogram operation by external subprogram call

In M198 or M-code for subprogram operation by external subprogram call (parameter No.6030), the backward movement is prohibited though the forward movement is enable.

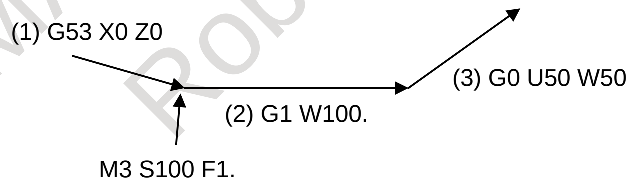
- Movement command and M,S,T-code

When M,S,T-codes and movement commands are in the same block, the timing outputting codes changes between in forward movement and backward movement. Therefore, M, S, T-codes should be commanded in backward movement after confirming that "DEN" signal is set to 1.

Example of executing the following programs with lathe system

```
O0001 ;
M5 S0 F0 ;
G53 X0 Z0 ; ............................................................................. (1)
G1 W100 M3 S100 F1. ; ................................................................ (2)
G0 U50. W50. ; ......................................................................... (3)
M2 ;
```

[Forward movement]



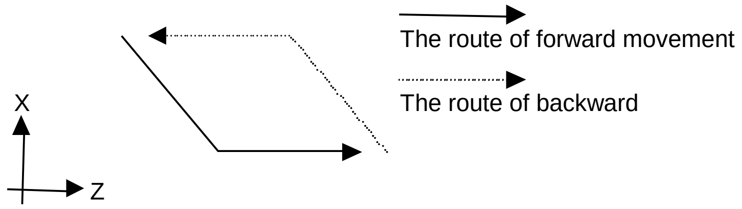
The block of (2) moves with M3 S100 F1.

[Backward movement]

The block of (2) moves with M5 S0 F1.

- Non linear interpolation type positioning

In the non-linear interpolation type positioning, the route is different between forward movement and backward movement.



Please use the interpolation type positioning to ward off danger. (Set the bit 1 (LPR) of parameter No. 1401 to 1)
If non-linear interpolation type positioning is used, direction change is prohibited at the end of movement along any axis.

- Threading

When bit 6 (HRG) of parameter No.6403 is set to 0, threading (G32,G76,G84,G88,G92)is always executed at 100% override speed. That is to say, a pulse generated by a manual handle is ignored in executing a threading block. In thread cutting cycle, the pulse is invalid at the time actually cutting thread, but the one are valid in the other movements.
When bit 6 (HRG) of parameter No.6403 is set to 1, it is possible to move backward and re-forward in the block of threading. However, threading cycle (G92) and multiple threading cycle (G76) are excluded.

> ⚠ **WARNING**
> When the parameter HRG is set to 1, detach the work if you execute the program including the threading because the synchronization of the spindle and the feed axis is not maintained at threading block. The machining of work is impossible.

- Rigid tapping (lathe system)

When bit 6 (HRG) of parameter No.6403 is set to 1, it is possible to move backward and re-forward in the block of rigid tapping. However, it is not possible to move backward by passing an on-off command of the rigid tapping mode.

> ⚠ **WARNING**
> When the parameter HRG is set to 1, detach the work if you execute the program including the rigid tapping. The machining of work is impossible.

- Tapping mode (G63)

When bit 6 (HRG) of parameter No.6403 is set to 1, it is possible to move backward and re-forward in the tapping mode.

> ⚠ **WARNING**
> When the parameter HRG is set to 1, detach the work if you execute the program including the tapping mode because the synchronization of the spindle and the feed axis is not maintained. The machining of work is impossible.

- Macro variable

In macro statement, the setting, operation, and so on of the macro variable is executed in only first forward movement. That is to say, the setting, operation, and so on of the macro variable is never executed in the block executing them once.

- Macro modal call

Manual handle retrace function is effective to macro modal call (G66,G66.1).

- Macro executor (execution macro)

Manual handle retrace function is effective while the execution macro of macro executor is called.

- Interruption type custom macro

Do not execute backward movement or re-forward movement after executing the interruption program of interruption type custom macro.

- Manual handle interrupt

Do not operate the manual handle interrupt simultaneously with manual handle retrace function.

- PMC axis control

The movement of PMC axis control cannot be controlled by this function.

- Time wait in a multi-path system

In a multi-path system, control is performed in such a way that during forward movement, the sum of the handle pulses input from the start and end of each block is memorized and during backward movement, the tool does not move to the previous block until as many handle pulses as the number of handle pulses memorized during forward movement are input. Any handle pulses input in an in-position check are also memorized as part of the pulses mentioned above, and if the speed of the handle (i.e., rate of movement along an axis) during forward movement differs from that during backward movement, the time required for an in-position check fluctuates, so that the sum of the handle pulses memorized during forward movement differs from the sum of the handle pulses during backward movement.
In this case, it can happen that even if a block ends during backward movement (the display of the remaining amount of travel becomes 0), the tool does not move to the previous block until the handle is rotated in the backward movement direction for the handle pulses memorized during forward movement. Also, at this time, the system is in the direction change prohibition state, and it is not possible to switch to forward movement until the tool continues backward movement to move to the previous block.

- Check of path unit by multi-path system

In the multi-path system, the program check of an arbitrary path is possible. In the path not to check, please select the mode excluding the MEM mode. If the bit 4 (HDMPH) of parameter No. 6400 is set to 1, it is possible to execute the check of forward movement, direction change and backward movement.
If waiting M code exist in the program, please set No-wait signal of two path NOWT <G0063.1>, or No-wait signal of three path NMWT<Gn063.9> to 1.

- Modal display

In the backward movement with manual handle, the modal display is updated according to the operation condition of the program.

- Modal information

In the backward movement with manual handle, the state of modal information is updated according to the operation condition of the program.

- Change in operation mode

When you change to EDIT mode during the checking mode, the backward movement and the re-forward movement cannot be executed in the blocks which have been already executed.

- ON/OFF of Manual Handle Retrace mode

When checking mode signal MMOD<Gn067.2> is set to 0 and handle available signal in checking mode MCHK<Gn067.3> is set to 0, the checking mode might not be turned off at once.
Basically, in the middle of block, the checking mode doesn't switch from ON to OFF or from OFF to ON. After the block is ended, the checking mode switches from ON to OFF or from OFF to ON.

- AI contour control

If the checking mode signal MMOD<Gn067.2> is 1, AI contour control mode is canceled. If AI contour control is canceled, forward movement and backward movement can be executed.

Furthermore, when checking mode signal MMOD<Gn067.2> is set to 1 in AI contour control mode, it is enabled a checking mode from the block that next buffering is done. Then, AI contour control is canceled. When bit 1 (FWD) of parameter No.6400 and bit 7 (HAI) of parameter No.6403 are 1, AI contour control is effective in the checking mode. Even if it is already AI contour control mode, the checking mode is effective.

- Multi-spindle control

During the backward movement, both TYPE-A and TYPE-B multi spindle control may not be operated exactly.

- Handle-synchronous feed

When manual handle retrace and a handle-synchronous feed are operated at the same time, forward movement and backward movement are done according to the operation of the first manual handle, and the feedrate of a handle-synchronous feed becomes effective.

- Execution of measurement G-code with the speed of override 100%

When the bit 6 (MGO) of parameter No. 6400 is set to 1, a handle pulse is invalid and it is always executed at a speed of 100% override while measurement G-code is executed. When the bit 6 (MGO) of parameter No. 6400 is set to 0, this function is invalid and a handle pulse is valid.
In the multi-path system, this function is not effective in the execution of another path and handle pulse is valid in another path. The measurement G code to which this function is effective is as follows.

1) G31 for skip
2) G31, G31P1, G31P2, G31P3, G31P4, G04, G04Q1, G04Q2, G04Q3 and G04Q4 for multistage skip
3) G31P99 and G31P98 for torque limit skip

When the measurement by G36 and G37 (G37.1 and G37.2. if bit 3 (G36) of parameter No.3405 is set to 1) of automatic tool compensation is executed, a handle pulse is invalid and it is always executed at a speed of 100% override regardless of setting the bit 6 (MGO) of parameter No. 6400. At the rapid traverse before measurement, handle pulse is valid.

When the bit 7 (SKF) of parameter No. 6200 is set to 0 and the bit 2 (SFN) of parameter No. 6207 is set to 0, handle pulse at G31 is invalid and it is always executed at a speed of 100% override regardless of setting the bit 6 (MGO) of parameter No. 6400.

When the bit 7 (MG4) of parameter No. 6400 is set to 1 and the multistage skip is enabled and the setting of parameter from No.6202 to No.6206 is enabled, the backward movement prohibition is enabled in G04 block for multistage skip.
The G code to which this function is effective is as follows.

1) G04, G04Q1, G04Q2, G04Q03 and G04Q4 for multistage skip

- Skip function, Multi-step skip function, High-speed skip function and Torque limit skip function

When bit 5 (HRF) of parameter No.6403 is set to 1, it is possible to move backward and re-forward in the block of G31.
During the backward movement or the re-forward movement, any skip signal and torque limit are ignored and it moves backward or re-forward on the path that moved first time forward movement.
The system variables #100151 to #100182 and #5061 to #5080 of custom macro are not updated during the backward movement and the re-forward movement.

- Reference position return check (G27)

The reference position return end signals ZP1 to ZP8 <Fn094> is not output in the backward movement.

- Polar coordinate interpolation
- Cylindrical interpolation
- 3-dimensional coordinate system conversion

It is not possible to move backward by passing an on-off block of these modes though it is possible to move backward and re-forward in these modes.

- Relation to another function

This function cannot coexist with the following functions

- Retrace

- Time constant for acceleration/deceleration after interpolation

A time constant for acceleration/deceleration after interpolation for manual handle retrace is disabled with the following functions.

- Spindle control with servo motor
- PMC axis control
- Threading (While executing the block of threading)
- Electronic gear box (EGB)
- Polygon turning
- Servo/spindle synchronous control
- Spindle of arbitrary speed threading

Message

The messages below will be displayed if direction change is prohibited during manual handle retrace operation.

Message	Description
PXX NO RVRS : PARAM OR SIGNAL	Direction change is prohibited because of the setting of bit 1 of parameter No. 6400 or signal G531.1.
PXX NO RVRS : IN AUXILIARY FUNCTION	Direction change is prohibited because an auxiliary function is being executed.
PXX NO RVRS : NON-LINEAR POSITIONING	Direction change is prohibited because non-linear positioning is in progress.
PXX NO RVRS : PROGRAM ENDING	Direction change is prohibited because of an M02 or M30 block.
PXX NO RVRS : FEED HOLD PROHIBITED	Direction change is prohibited because feed hold is prohibited due to, for example, threading.
PXX NO RVRS : ACC AFTER INTERPOLATION	Direction change is prohibited because an axis stop is awaited due to acceleration/deceleration after interpolation.
PXX NO RVRS : IN NO RVRS FUNCTION	Direction change is prohibited because of a modal code that does not allow direction change, such as polygon, balance cut, and 3-dimensional coordinate conversion.
PXX NO RVRS : WAITING M CODE	Direction change is prohibited because a waiting M code is being executed.
PXX NO RVRS : IN NO RVRS BLOCK	Direction change is prohibited because a block that does not allow backward movement is being executed.
PXX NO RVRS	Direction change is prohibited for another reason.
PXX IS TOP OF THE PROGRAM	The reverse movement of the program has ended.

XX: Path number 1 to 10

NOTE

If direction change is prohibited on two or more paths, the message for the path on which direction change is prohibited last will be displayed.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6400	MG4	MGO	RVN	HMP	MC8	MC5	FWD	RPO

[Input type] Parameter input
[Data type] Bit path

#0 RPO With the manual handle retrace function, the rapid traverse rate is clamped, assuming that:
0: An override of 10% is used.
1: An override of 100% is used.

#1 FWD With the manual handle retrace function, program execution can be performed:
0: In both forward and backward directions.
1: In the forward direction only. Execution in the backward direction is not permitted.

#2 MC5
#3 MC8 These parameters set the number of M code groups and the number of M codes per group. (See explanations of parameters Nos. 6411 to 6490.)

MC5	MC8	M code group setting
0	0	Standard (20 groups of four)
1	0	16 groups of five
0	1	10 groups of eight

When 16 groups of five are used, the meanings of parameters are changed as follows:
Group A No. 6411(1) to No. 6415(5)
Group B No. 6416(1) to No. 6420(5)
:
Group P No. 6486(1) to No. 6490(5)
When 10 groups of eight are used, they are changed as follows:
Group A No. 6411(1) to No. 6418(8)
Group B No. 6419(1) to No. 6426(8)
:
Group J No. 6483(1) to No. 6490(8)

#4 HMP When inversion or backward movement is inhibited in other paths:
0: Inversion or backward movement is not inhibited for the currently executed path.
1: Inversion or backward movement is inhibited also for the currently executed path.

#5 RVN When the manual handle retrace function is used, M codes other than grouped M codes:
0: Do not disable backward movement.
1: Disable backward movement.

When this parameter is set to 1, M codes other than grouped M codes disable backward movement in general. Exceptionally, however, the following M codes allow backward movement:
1. Subprogram call based on M98/M99
2. Subprogram call based on an M code
3. Macro call based on an M code
4. Waiting M code
5. M0

#6 MGO When the manual handle retrace function is used, handle pulses during execution of a G code related to measurement are:
0: Valid.
1: Invalid. A speed with an override of 100% is used for execution at all times.

#7 MG4 In the manual handle retrace function, for blocks for which multi-step skip G04 is enabled (when the multi-step skip is used, and the settings of parameters Nos. 6202 to 6206 are valid):
0: Backward movement is not prohibited.
1: Backward movement is prohibited.

	#7	#6	#5	#4	#3	#2	#1	#0
6401	STO	HST				CHS		

[Input type] Parameter input
[Data type] Bit path

#2 CHS In manual handle retrace:
0: The status is displayed if the following conditions are all satisfied:
(1) Bit 6 (HST) of parameter No. 6401, which specifies whether to enable or disable status display, is set to 1.
(2) Check mode output signal MMMOD<Fn091.3> is set to 1.
1: The status is displayed if the following conditions are all satisfied:
(1) Bit 6 (HST) of parameter No. 6401, which specifies whether to enable or disable status display, is set to 1.
(2) Cycle start lamp signal STL<Fn000.5> is set to 1.
(3) Checking mode input signal MMOD<Gn067.2> is set to 1.
(4) Handle input signal MCHK<Gn067.3> is set to 1 in the checking mode.

#6 HST When the manual handle retrace function is used, the time display field on the status display line of the CNC screen:
0: Does not display status.
1: Displays status.

#7 STO In the manual handle retrace function, the timing for outputting an S code and T code during backward movement is:
0: Different from the timing during forward movement:
1: The same as during forward movement.

	#7	#6	#5	#4	#3	#2	#1	#0
6402			MWR					

[Input type] Parameter input
[Data type] Bit path

#5 MWR When the manual handle retrace function is used, for a handle operation placed in the wait state by a waiting M code during backward movement:
0: Inversion is prohibited.
1: Inversion is permitted.

	#7	#6	#5	#4	#3	#2	#1	#0
6403	HAI	HRG	HRF					

[Input type] Parameter input
[Data type] Bit path

#5 HRF In skip function, multi-step skip function, high-speed skip function and torque limit skip function, with the manual handle retrace function, program execution in the backward and re-forward direction :
0: Cannot be performed.
1: Can be performed.

> **NOTE**
> 1 Any skip signal and torque limit are ignored and it moves backward and re-forward in the distance that moved first time forward movement during the backward movement and the re-forward movement.
> 2 The system variables #100151 to #100182 and #5061 to #5080 of custom macro are not updated during the backward movement and the re-forward movement.

#6 HRG In rigid tapping (lathe system) and threading, with the manual handle retrace function, program execution in the backward and re-forward direction :
0: Cannot be performed.
1: Can be performed.

> ⚠ **WARNING**
> 1 When the parameter HRG is set to 1, detach the work if you execute the program including the rigid tapping. The machining of work is impossible.
> 2 When the parameter HRG is set to 1, detach the work if you execute the program including the threading because the synchronization of the spindle and the feed axis is not maintained at the threading block. The machining of work is impossible.

#7 HAI In manual handle retrace mode, AI contour control is:
0: Disabled.
1: Enabled.

> **NOTE**
> This parameter is enabled when bit 1 (FWD) of parameter No.6400 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
6404								HMD

[Input type] Parameter input
[Data type] Bit

#0 HMD In manual handle retrace, when inversion or backward movement is inhibited:
0: Cause is not displayed.
1: Cause is displayed.

6405	**Override value (equivalence) for clamping the rapid traverse rate used with the manual handle retrace function**

[Input type] Parameter input
[Data type] Word path

[Unit of data] %
[Valid data range] 0 to 100
This parameter sets an override value (equivalence) for clamping the rapid traverse rate used with the manual handle retrace function. If a value greater than 100 is set in parameter No. 6405, the rapid traverse rate is clamped to an override of 100%. This function is invalid if 0 is set in parameter No. 6405. In this case, the setting of bit 0 (RPO) of parameter No. 6400 is used.

6410	**Travel distance per pulse generated from the manual pulse generator**

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 0 to 100
Set the travel distance per pulse generated from the manual pulse generator in terms of the override value.
The distance traveled by the machine when the manual handle is actually turned can be found by the following expression:
[Specified speed] × [Handle magnification] × ([Setting of this parameter]/100) × (8/60000) (mm or inch)

[Example] When a specified feedrate is 30mm/min, the manual handle magnification is 100, and parameter No. 6410 is set to 1, the travel distance per pulse generated from the manual pulse generator is calculated as follows:
[Travel distance per pulse]=30[mm/min] × 100 × (1/100) × (8/60000)[min]= 0.004mm
When this parameter is set to 0, the amount of the movement becomes 0. Moreover, it is equivalent to 100 in case of 100 or more.

6411	**M code of group A in manual handle retrace (1)**
to	**to**
6414	**M code of group A in manual handle retrace (4)**
6415	**M code of group B in manual handle retrace (1)**
to	**to**
6418	**M code of group B in manual handle retrace (4)**
6419	**M code of group C in manual handle retrace (1)**
to	**to**
6422	**M code of group C in manual handle retrace (4)**
6423	**M code of group D in manual handle retrace (1)**
to	**to**
6426	**M code of group D in manual handle retrace (4)**
6427	**M code of group E in manual handle retrace (1)**
to	**to**
6430	**M code of group E in manual handle retrace (4)**
6431	**M code of group F in manual handle retrace (1)**
to	**to**
6434	**M code of group F in manual handle retrace (4)**
6435	**M code of group G in manual handle retrace (1)**
to	**to**
6438	**M code of group G in manual handle retrace (4)**
6439	**M code of group H in manual handle retrace (1)**
to	**to**
6442	**M code of group H in manual handle retrace (4)**

6443	M code of group I in manual handle retrace (1)
to	to
6446	M code of group I in manual handle retrace (4)

6447	M code of group J in manual handle retrace (1)
to	to
6450	M code of group J in manual handle retrace (4)

6451	M code of group K in manual handle retrace (1)
to	to
6454	M code of group K in manual handle retrace (4)

6455	M code of group L in manual handle retrace (1)
to	to
6458	M code of group L in manual handle retrace (4)

6459	M code of group M in manual handle retrace (1)
to	to
6462	M code of group M in manual handle retrace (4)

6463	M code of group N in manual handle retrace (1)
to	to
6466	M code of group N in manual handle retrace (4)

6467	M code of group O in manual handle retrace (1)
to	to
6470	M code of group O in manual handle retrace (4)

6471	M code of group P in manual handle retrace (1)
to	to
6474	M code of group P in manual handle retrace (4)

6475	M code of group Q in manual handle retrace (1)
to	to
6478	M code of group Q in manual handle retrace (4)

6479	M code of group R in manual handle retrace (1)
to	to
6482	M code of group R in manual handle retrace (4)

6483	M code of group S in manual handle retrace (1)
to	to
6486	M code of group S in manual handle retrace (4)

6487	M code of group T in manual handle retrace (1)
to	to
6490	M code of group T in manual handle retrace (4)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 9999

Set a group of M codes output during backward movement.
For backward movement for an M code, the modal M code in the same group set by the parameter is output.
The first M code in each group is set as the default.
When the number of M codes in a group is 3 or less, set the parameter corresponding to an unused M code to 0.
For backward movement for "M0", "M0" is output regardless of which M code is set for the parameter. 0 set in the parameter is ignored.

For an M code which is not set in any group by any of the above parameters, the M code for forward movement is output.
With these parameters, an M code in the same group can be output in backward movement only when the M code is the first M code in each block. When a block contains two or more M codes, the same M codes as output in forward movement are output as a second M code and up.

> **NOTE**
> The above explanation of M code groups applies to the standard settings. The number of M codes in each group and the number of M code groups vary depending on the settings of bits 2 (MC5) and 3 (MC8) of parameter No. 6400.

6495	Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis for manual handle retrace

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Specify a time constant used for acceleration/deceleration in rapid traverse for manual handle retrace. For an axis with 0 set in this parameter, a time constant for acceleration/deceleration of parameter (No.1620) is enabled.

6496	Time constant T_2 used for bell-shaped acceleration/deceleration in rapid traverse for each axis for manual handle retrace

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 1000

Specify time constant T_2 used for bell-shaped acceleration/ deceleration in rapid traverse for each axis for manual handle retrace. For an axis with 0 set in this parameter, a time constant for acceleration/deceleration of parameter (No.1621) is enabled.

6497	Time constant of acceleration/deceleration in cutting feed for each axis for manual handle retrace

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in cutting feed, bell-shaped acceleration/deceleration after interpolation or linear
acceleration/deceleration after interpolation in cutting feed for each axis for manual handle retrace. Which type to use is selected with bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610. Except for special applications, the same time constant must be set for all axes in this parameter. If the time constants set for the axes differ from each other, proper straight lines and arcs cannot be obtained. For an axis with 0 set in this parameter, a time constant for acceleration/deceleration of parameter (No.1622) is enabled.

18060	M code that prohibits backward movement

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 999

When an M code that prohibits backward movement is specified during backward movement, backward movement of blocks before the M code is prohibited. In this case, backward movement prohibition signal MRVSP<Fn091.2> is output.

The M code that prohibits backward movement is not output to the PMC as an M code. As the M code that prohibits backward movement, set an M code that is not used by auxiliary functions and macros.

18065	M code 1 that prohibits backward movement and is output as an M code
18066	M code 2 that prohibits backward movement and is output as an M code

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 999

When an M code that prohibits backward movement is specified during backward movement, backward movement of blocks before the M code is prohibited. In this case, backward movement prohibition signal MRVSP<Fn091.2> is output.

Such M codes that prohibit backward movement are output to the PMC as M codes. As the M codes that prohibit backward movement, set M codes that are not used by auxiliary functions and macros.

Signal

Checking mode signal MMOD<Gn067.2>

[Classification] Input signal
[Function] The status of the manual handle retrace mode is changed.
[Operation] In case that the automatic operation is executed in the MEM mode when this signal is 1, the mode of the manual handle retrace becomes "ON".
In the MDI operation, the mode becomes "OFF".
Moreover, when this signal is set to 1 in the automatic operation in the MEM mode, the mode will become "ON" from the block executed next.
When this signal is set to 0 in the automatic operation, the mode will become "OFF" from the block executed next.

Handle available signal in checking mode MCHK<Gn067.3>

[Classification] Input signal
[Function] Enabling/disabling of the manual handle operation in the manual handle retrace mode is changed.
[Operation] When this signal is set to 1 in the manual handle retrace mode, the program is executed with the speed corresponding to the rotating speed of a manual handle. Moreover, when the manual handle is rotated to the opposite direction, the program is executed to the opposite direction.
When this signal is set to 0, the program is executed with the speed instructed in the program to the direction of the forward movement as same as a usual automatic operation.
When this signal is changed from 1 to 0 during the program, the program is executed as well.

Forward movement prohibition signal FWSTP<Gn531.0>

[Classification] Input signal

[Function] The forward movement in the manual handle retrace mode is prohibited.

[Operation] When this signal is set to 1 in the manual handle retrace mode, the forward movement of the program is prohibited. If this signal is set to 1 during the status of direction change prohibition or the status of the backward movement prohibition, both of the forward movement and the backward movement cannot be done.

Reverse movement prohibition signal MRVM<Gn531.1>

[Classification] Input signal

[Function] The backward movement in the manual handle retrace mode is prohibited.

[Operation] When this signal is set to 1 in the manual handle retrace mode, the backward movement of the program is prohibited. The backward movement is not done and the program stops even if a manual handle is rotated to the opposite direction.

Check mode confirmation signal MMMOD<Fn091.3>

[Classification] Output signal

[Function] This signal indicates the manual handle retrace mode.

[Operation] This signal becomes 1 when the manual handle retrace mode is "ON".
When the manual handle retrace mode is turned off, this signal becomes 0.

Reverse movement signal MRVMD<Fn091.0>

[Classification] Output signal

[Function] This signal indicates that program is being backward movement.

[Operation] This signal becomes 1 in the manual handle retrace mode when the backward movement is executed.
It becomes 0 during the forward movement and the re-forward movement.

Direction change prohibition signal MRNCHG<Fn091.1>

[Classification] Output signal

[Function] This signal indicates that direction change is being prohibited.

[Operation] It becomes 1 when the direction to which the program is executed cannot be changed(The status of the direction change prohibition) in the manual handle retrace mode.
When the backward movement of the program can be done, it becomes 0.

Reverse movement prohibition signal MRVSP<Fn091.2>

[Classification] Output signal

[Function] This signal indicates that the backward movement is being prohibited.

[Operation] It becomes 1 when the backward movement can be executed no more (The status of the backward movement prohibition) from a certain block in the manual handle retrace mode.
When the backward movement of the program can be executed, it becomes 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn067					MCHK	MMOD		
Gn531							MRVM	FWSTP

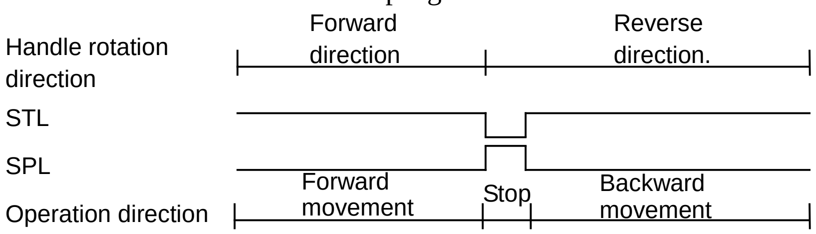
	#7	#6	#5	#4	#3	#2	#1	#0
Fn091					MMMOD	MRVSP	MNCHG	MRVMD

Notes

1) In manual handle retrace function, when checking mode signal MMOD<Gn067.2> is set to 0 and handle available signal in checking mode MCHK<Gn067.3> is set to 0, the check mode might not be turned off at once.

Basically, in the middle of block, the check mode doesn't switch from ON to OFF or from OFF to ON. After the block is ended, the check mode switches from ON to OFF or from OFF to ON.

2) When the rotate direction of the manual handle is changed, the timing chart of the cycle start lamp signal STL<Fn000.5> and the feed hold lamp signal SPL<Fn000.4> are shown in the figure below.



4) When manual handle retrace function is executed at the same time in two or more paths, some gaps might be caused in operation between each paths by repetition of forward movement and backward movement or rotational speed of manual handle. Waiting M codes are used for the synchronized block between each paths.

5.3.6 Auxiliary Function Output Block Reverse Movement for Manual Handle Retrace

Overview

This function enables reverse movement during manual handle retrace even if a move command and an auxiliary function (M/S/T/B code) are specified in the same block.

Explanation

Reverse movement is enabled for an auxiliary function output block if all the conditions and sequences described below are met.

<1> The function is effective when bit 0 (ADC) of parameter No. 6401 (auxiliary function output block reverse movement enable parameter) is set to 1.

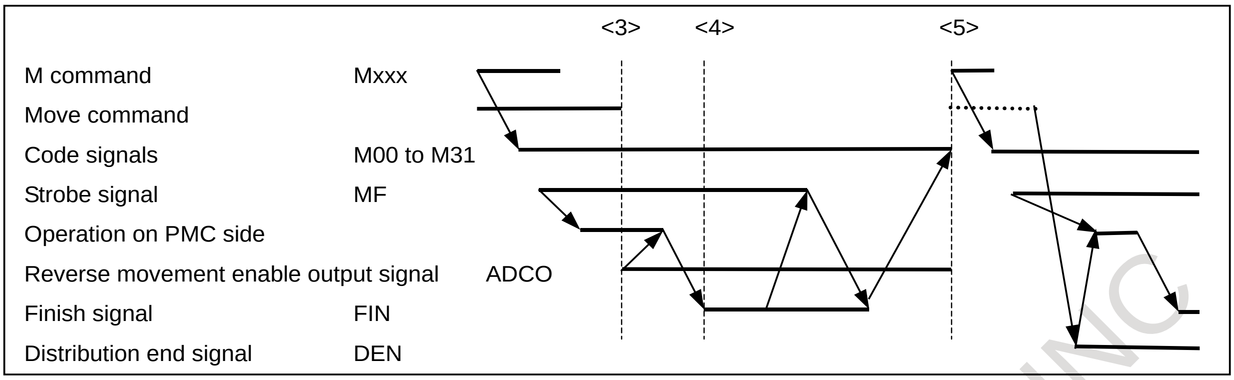
<2> The function is effective when a move command and an auxiliary function (M/S/T/B code) are specified in the same block.

<3> If the rotation of the manual pulse generator is reversed when all the conditions mentioned in <1> and <2> are met, the NC sets auxiliary function output block reverse movement enable output signal ADCO<Fn091.5> to 1.

<4> When auxiliary function output block reverse movement enable output signal ADCO<Fn091.5> is set to 1, the PMC performs the corresponding operation with a ladder language program. After that, set finish signal FIN<Gn004.3> to 1. This enables the NC to move the axis in reverse direction (move it backward or move it forward again).

<5> When finish signal FIN<Gn004.3> is set to 1, the NC sets ADCO<Fn091.5> to 0 and starts to move the axis in reverse direction (move it backward or move it forward again).
As for parameter-grouped M codes or S or T codes, those of the previous one block are output. As for M codes that are not grouped by parameter or B codes, the same codes as for the forward movement are also output for the reverse movement.

The figure below presents a timing chart depicting this process. The chart shows an example of M codes.
This example describes how an auxiliary function is executed after waiting for the move command to complete (distribution end signal DEN) following the reverse movement.
It is also possible to execute an auxiliary function without waiting for the move command to complete after the reverse movement.



Signal

Auxiliary function output block reverse movement enable output signal ADCO<Fn091.5>

[Classification] Output signal

[Function] Indicates that the manual handle has reversed during manual handle retrace in a block where a move command and an auxiliary function (M/S/T/B code) are specified together.

[Output cond.] This signal is set to 1 if all the conditions described below are met.

1) Bit 0 (ADC) of parameter No. 6401 (auxiliary function output block reverse movement parameter) is set to 1.
2) A move command and an auxiliary function (M/S/T/B code) are specified in the same block.
3) Finish signal FIN is being awaited after the rotation of the manual pulse generator has reversed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn091			ADCO					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6401								ADC

[Input type] Parameter input
[Data type] Bit path

#0 ADC If a move command and an auxiliary function (M/S/T/B code) are specified in the same block when the manual handle retrace function is used, the block:

0: Disables reverse movement.
1: Does not disable reverse movement.

Notes

NOTE

1 When a single auxiliary function is specified individually, carry out the regular auxiliary function completion sequence. Reverse movement becomes possible after moving to the next (or previous) block.

2 If a move command and an auxiliary function are specified in the same block and if no travel distance remains after movement, carry out the regular auxiliary function completion sequence. Reverse movement becomes possible after moving to the next (or previous) block.

NOTE

3 To enable reverse movement using this function when an M code other than grouped M codes and a move command are specified in the same block, set bit 5 (RVN) of parameter No. 6400 to 0.
4 This function is not supported for M02 or M30.
5 If an auxiliary function is specified in a block having no move command, such as G70, carry out the regular auxiliary function completion sequence.
Reverse movement becomes possible after moving to the next (or previous) block.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Manual handle retrace

5.3.7 Manual Handle Retrace Function for Multi-path

Overview

In the manual handle retrace function for multi-path systems, when the operation of re-forward movement is performed, the movement timing of all-path movement can be made identical to that of forward movement.

Explanation

In the manual handle retrace function for multi-path systems, when a certain path stopped with the backward movement prohibited and the other paths continue the backward movement, if re-forward movement is performed after that, all paths start the re-forward movement immediately, so the timing of block movement of each path is different from the original forward movement.
In this function, during re-forward movement, the path that stopped with backward movement prohibited doesn't start re-forward movement immediately. After other paths reach the position the path stopped with backward movement prohibited, the re-forward movement of this path is performed. Therefore, in multi-path systems, it is possible to perform re-forward movement in which the timing of block movement of each path is made identical to that of forward movement.
This function is enabled when bit 1 (RTW) of parameter No.18000 is set to 1.

For information on basic movement (forward, backward, re-forward) of the manual handle retrace function and the settings, refer to Subsection, "Manual Handle Retrace".

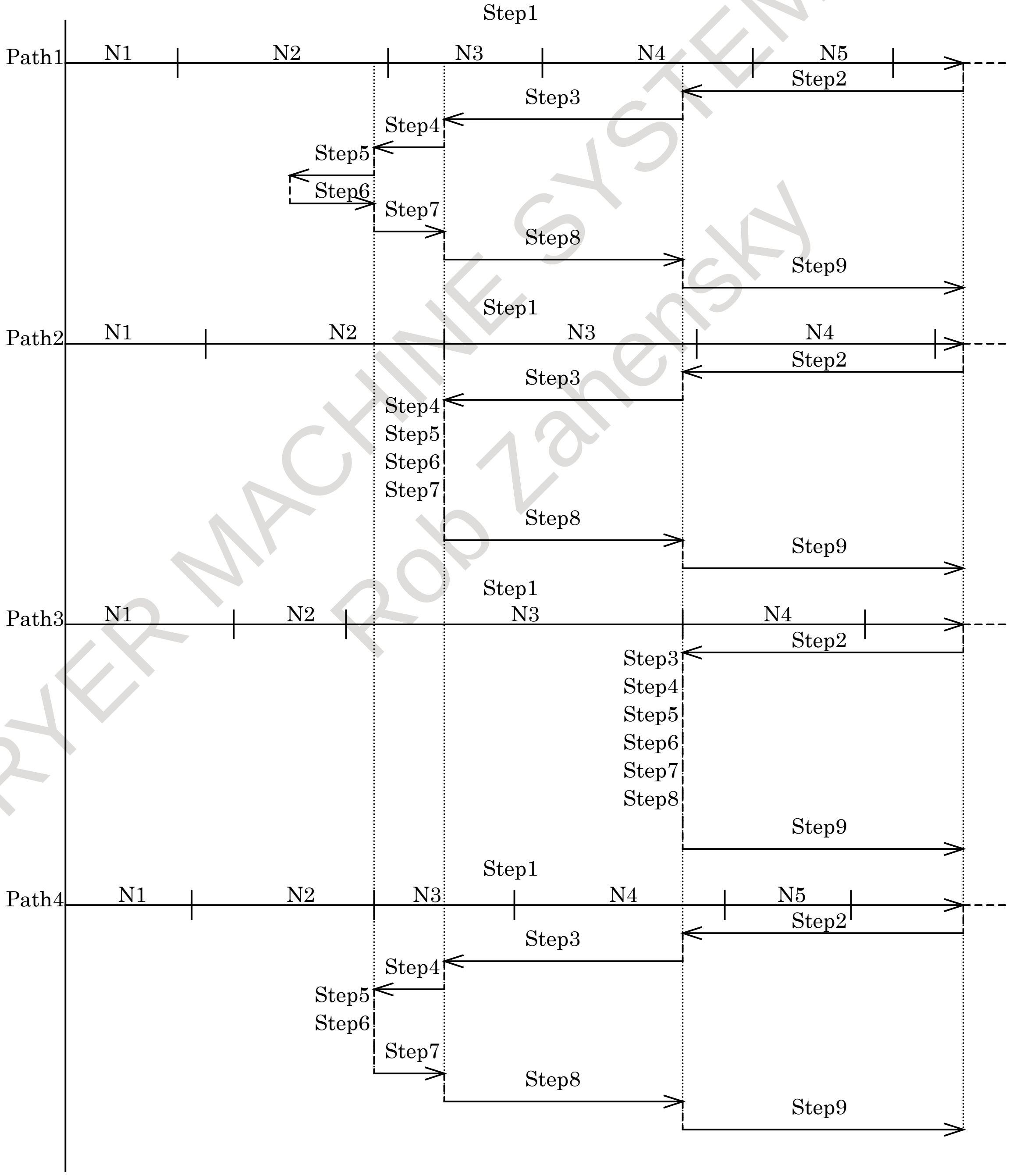
NOTE

Even when this function is enabled, the timing of block movement of each path may differ slightly due to the repetition of forward and backward movement and the rotation speed of the manual handle. Therefore, when synchronization is necessary, use the waiting M code.

Example) Re-forward movement of manual handle retrace in a 4-path system

Path1	Path2	Path3	Path4
O1001	O1001	O1001	O1001
N1 G1 X5.0 F1	N1 G1 Z6.0 F1	N1 G1 X7.0 F1	N1 G1 Z5.0 F1
N2 X12.0	N2 G31 Z14.0	N2 X10.0	N2 G31 Z11.0
N3 X22.0	N3 Z28.0	N3 G31 X27.0	N3 Z18.0
N4 X30.0	N4 Z36.0	N4 X33.0	N4 Z29.0
N5 X34.5	N5 Z50.0	N5 X50.0	N5 Z32.0
N6 X50.0	M30	M30	N6 Z50.0
M30			M30

When the above programs are operated in each path, the operation states of individual steps (in forward, backward, and re-forward movement) are shown below.



STEP	1	2	3	4	5	6	7	8	9
Command state	Forward	Backward	Backward	Backward	Backward	Re-forward	Re-forward	Re-forward	Re-forward
Execution state in Path1	Forward	Backward	Backward	Backward	Backward	Re-forward	Re-forward	Re-forward	Re-forward
Execution state in Path2	Forward	Backward	Backward	Stop	Stop	Wait P1	Wait P1,4	Re-forward	Re-forward
Execution state in Path3	Forward	Backward	Stop	Stop	Stop	Wait P1	Wait P1,4	Wait P1,2,4	Re-forward
Execution state in Path4	Forward	Backward	Backward	Backward	Stop	Wait P1	Re-forward	Re-forward	Re-forward

However, when bit 4 (HMP) of parameter No. 6400 is 0 (even if a path is prohibited from changing movement direction, the other paths can still change it), if operation is performed in the condition shown in the example below, re-forward movement may not keep the same motion as in forward movement.

Example)

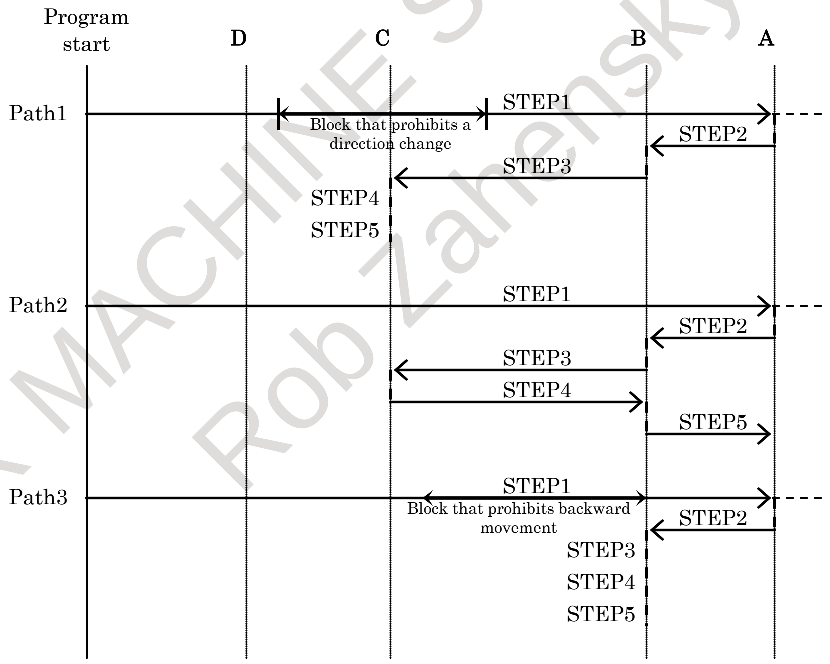
Conditions of the programs of the individual paths

Path1... In backward movement, there is a block that prohibits a change to forward movement.

Path2... In backward movement, there is neither block that prohibits backward movement nor block that prohibits a change to forward movement.

Path3... In backward movement, there is a block that prohibits backward movement.

In the above conditions, the following operation is performed.

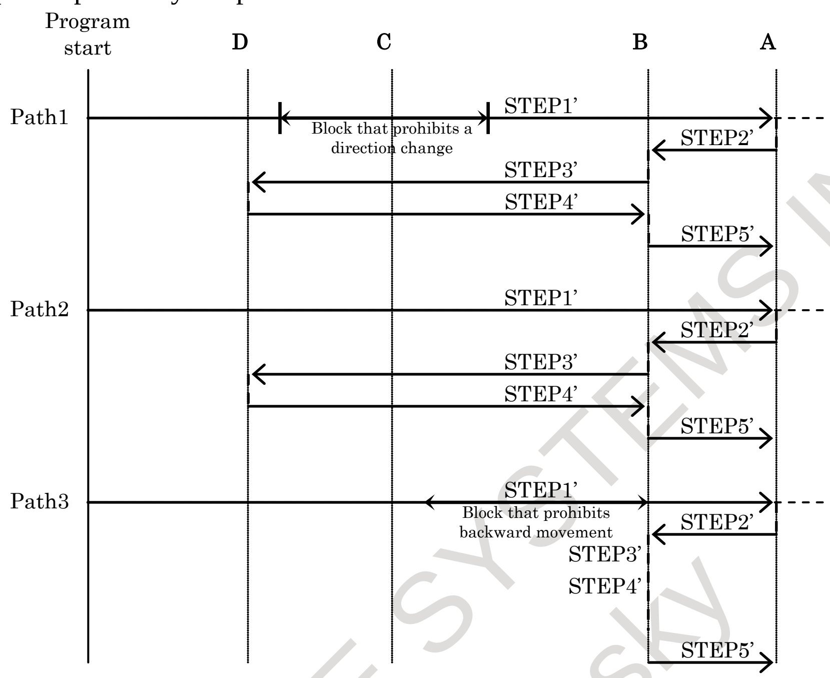


STEP1: Forward movement is performed to position A in the figure.

STEP2, STEP3: Backward movement is performed to position C in the figure. Path 3 is prohibited from moving backward at position B.

STEP4, STEP5: Re-forward movement is performed to position A in the figure. Since path 1 is prohibited from changing its direction, only path 2 performs re-forward movement. Even when path 2 performs re-forward movement to position B, path 3 does not start re-forward movement.

To keep synchronization in re-forward movement, perform backward movement of path 1 until path 1 passes by block D, which prohibits a direction change, and start re-forward movement of path 1.
In this case, if bit 4 (HMP) of parameter No.6400 is set to 1 (if a path is prohibited from changing its movement direction, the other paths cannot also change their direction), the other paths cannot change their direction until path 1 passes by the position D.



STEP1': Forward movement is performed to position A in the figure.
STEP2',STEP3': Backward movement is performed to position D in the figure. Path 3 is prohibited from moving backward at position B.
STEP4',STEP5: Re-forward movement is performed to position A in the figure. When path 1 and path 2 perform re-forward movement to position B, path 3 starts re-forward movement.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6400				HMP				

[Input type] Parameter input
[Data type] Bit path

#4 HMP When inversion or backward movement is inhibited in other paths:
0: Inversion or backward movement is not inhibited for the currently executed path.
1: Inversion or backward movement is inhibited also for the currently executed path.

	#7	#6	#5	#4	#3	#2	#1	#0
18000							RTW	

[Input type] Parameter input
[Data type] Bit

#1 RTW At the start of a re-forward movement operation of the manual handle retrace function in a multi-path system,
0: The re-forward movement operation is performed immediately on each path.
1: Those paths for which reverse movement are prohibited are synchronized at the stop position.

5.3.8 Extension of the Manual Handle Retrace Function

Overview

In manual handle retrace function, the following operations that were impossible so far become available.
(1) Forward movement of rigid tapping
(2) Forward movement of threading
(3) Forward movement of PMC axis control
(4) Backward movement of the orientation by the G00 command on the Cs contour control axis
(5) Backward movement of polygon turning with two spindles
(6) Backward movement of balance cutting
This function is included in the multi-path manual handle retrace function.

Explanation

- Rigid tapping

- When the bit 0 (HRA) of parameter No.6403 is set to 0 (Conventional specification)
  (1) When rigid tapping starts during forward movement, handle operation is disabled and rigid tapping is executed at a speed of 100% override.
  (2) Backward movement cannot be performed in the rigid tapping mode. In addition, when the rigid tapping block is reached after rigid tapping is completed in forward movement, backward movement cannot be done any more.
- When the bit 0 (HRA) of parameter No.6403 is set to 1 (Extensional specification)
  (1) In tapping during forward movement, the operation is performed at a speed according to handle operation.
  (2) Backward movement in rigid tapping is the same movement as in the case where bit 0 (HRA) of parameter No. 6403 is 0.

- Thread cutting

- When the bit 0 (HRA) of parameter No.6403 is set to 0 (Conventional specification)
  (1) When the threading block is reached during forward movement, handle operation become disabled and the operation is performed at a speed of 100% override.
  (2) Backward movement cannot be performed during threading. In addition, when the threading block is reached during backward movement after threading is completed in forward movement, backward movement cannot be done any more.
- When the bit 0 (HRA) of parameter No.6403 is set to 1 (Extensional specification)
  (1) In threading during forward movement, the operation is performed at a speed according to handle operation.
  (2) Backward movement in threading is the same movement as in the case where bit 0 (HRA) of parameter No. 6403 is 0.

> ⚠ **WARNING**
> In rigid tapping or threading during manual handle retrace, the workpiece cannot be actually machined.

- PMC axis control

- When the bit 1 (HRB) of parameter No.6403 is set to 0 (Conventional specification)
  (1) The PMC axis operates independently of handle operation.
- When the bit 1 (HRB) of parameter No.6403 is set to 1 (Extensional specification)
  (1) In PMC axis control during forward movement, the PMC axis is operated at a speed according to handle operation but, backward movement cannot be performed.

NOTE

1 Even when the axis specified by the program moves backward, the axis specified by PMC axis control cannot move backward. Therefore, the relative position between the axis specified by the program and the axis specified by PMC axis control changes when backward movement is performed while the axis that is specified by PMC axis control is operating. In this case, it is necessary to add the following PMC ladder process.
   - When the axis specified by PMC axis control is operating, prohibit the backward movement of the axis specified by the program using reverse movement prohibition signal MRVM <Gn531.1>.

2 Alarm PS0130 occurs when one of the following operations is executed.
   (1) In the backward movement, when PMC axis control is specified for the axis specified by the program in the previous block.
   (2) During PMC axis control, when a direction change is performed for the axis specified by the program in the previous block.

3 In the skip (03h), continuous feed (06h), speed command (10h), torque control (11h), and auxiliary function (12h,13h,14h) of PMC axis control, even if bit 1 (HRB) of parameter No. 6403 is set to 1, the axis operates independently of handle operation.

⚠ **WARNING**

During the manual handle retrace, if a reset is made when a command by PMC axis control is not completed, the command by the program stops, but the command by PMC axis control continues. In this case, even if bit 1 (HRB) of parameter No. 6403 is set to 1, the command by PMC axis control operates independently of handle operation. Make a reset after confirming that the command by the PMC axis control is completed.

- Orientation of Cs contour control axis

- When the bit 2 (HRC) of parameter No.6403 is set to 0 (Conventional specification)
  (1) Backward movement cannot be performed in orientation movement by the first G00 command after the Cs contour control mode is turned on. Therefore, when the above G00 command is reached during backward movement, backward movement cannot be done any more.
  (2) Backward movement can be performed in the reference point return command (G28) of the Cs contour control axis, but the processing proceeds to the previous block without performing orientation movement.
- When the bit 2 (HRC) of parameter No.6403 is set to 1 (Extensional specification)
  (1) Backward movement can be performed in the orientation operation by the G00 command, but the processing proceeds to the previous block without performing orientation movement as in G28 command.
  (2) The reference point return command (G28) of the Cs contour control axis performs the same movement as in the case where bit 2 (HRC) of parameter No. 6403 is 0.

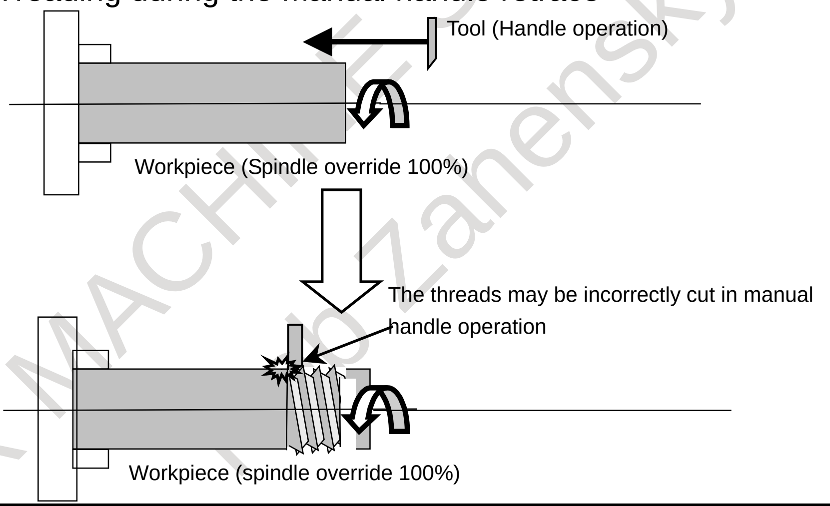
⚠ **WARNING**

The workpiece cannot be actually machined because the orientation movement of G00 and G28 is not executed during backward movement in Cs contour control.

- Polygon machining with two spindles

- When the bit 3 (HRD) of parameter No.6403 is set to 0 (Conventional specification)
  (1) Axes other than the polygonal synchronization axes operate at the speed according to handle operation even in the polygon machining mode. However, backward move cannot be performed in the block specified in the polygonal machining mode.
  (2) When the polygon machining block (G50.2) is reached in backward movement after the polygon machining is completed in forward movement, backward movement cannot be done any more.
- When the bit 3 (HRD) of parameter No.6403 is set to 1 (Extensional specification)
  (1) It is possible to move backward or re-forward in the block specified in the polygonal machining mode. When backward movement is performed during polygon machining, however, if the polygon machining-on block (G51.2) is reached, backward movement cannot be done any more.
  (2) The operation when the polygon machining block (G50.2) is reached in backward movement after the polygon machining is completed in forward movement the same as in the case where bit 3 (HRD) of parameter No. 6403 is 0.

> ⚠ **WARNING**
> In the threading and polygon machining between two spindles, the spindle operates at a speed of override 100% instead of the speed according to handle operation. Therefore, the workpiece cannot be actually machined.
>
> (Example) Treading during the manual handle retrace

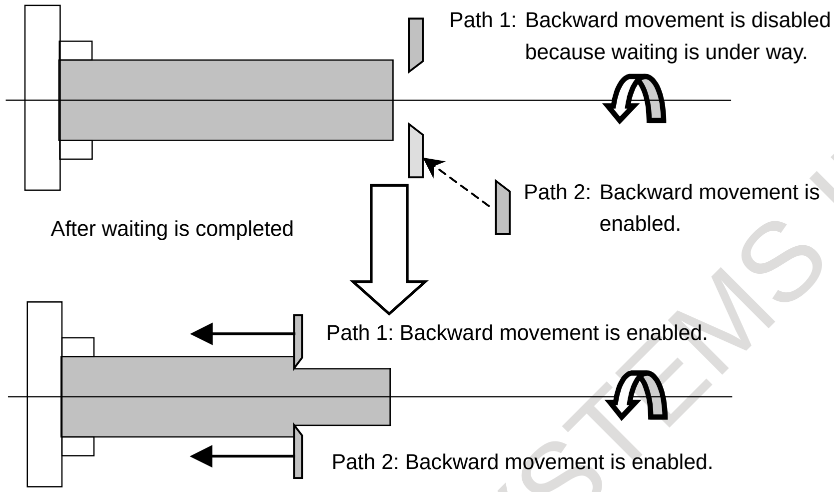


- Balanced cutting

- When the bit 4 (HRE) of parameter No.6403 is set to 0 (Conventional specification)
  (1) In the balance cutting mode, axes operate at the speed according to handle operation, but backward movement cannot be performed.
  (2) When the balance cutting block (G69) is reached in backward movement after the balance cutting is completed in forward movement, backward movement cannot be done any more.
- When the bit 4 (HRE) of parameter No.6403 is set to 1 (Extensional specification)
  (1) It is possible to move backward or re-forward in the block specified in the balance cutting mode. When backward movement is performed in the balance cutting mode, if the balance cut-on block (G68) is reached, backward movement cannot be done any more.
  (2) The operation when the balance cutting block (G69) is reached in the backward movement after the balance cutting was completed in forward movement is the same operation as in the case where bit 4 (HRE) of parameter No. 6403 is 0.

NOTE

1 In the balance cutting mode, waiting is done so that the block starts at the same time in each path. Backward movement cannot start during the waiting. After the waiting is completed and the next block starts, backward movement becomes possible.



2 In all paths that uses balance cutting, set bit 4 (HRE) of parameter No. 6403 to 1.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6403				HRE	HRD	HRC	HRB	HRA

[Input type] Parameter input
[Data type] Bit path

#0 HRA In rigid tapping and thread cutting, with the manual handle retrace function, program execution in the forward direction:
0: Cannot be performed.
1: Can be performed.

#1 HRB In PMC axis control, with the manual handle retrace, program execution in the forward direction:
0: Cannot be performed.
1: Can be performed.

#2 HRC During orientation operation according to G00 for a Cs contour control axis, with the manual handle retrace function, program execution in the backward direction:
0: Cannot be performed.
1: Can be performed.

#3 HRD In polygon machining with two spindles, with the manual handle retrace function, program execution in the backward direction:
0: Cannot be performed.
1: Can be performed.

#4 HRE In balanced cutting, with the manual handle retrace function, program execution in the backward direction:
0: Cannot be performed.
1: Can be performed.

5.4 MANUAL ABSOLUTE ON/OFF

Overview

Whether the distance by which the tool is moved by manual operation (such as jog feed or manual handle feed) is reflected in the absolute coordinate system can be selected.

- When manual absolute turns on

When manual intervention is performed during automatic operation, the distance by which the tool is moved by manual intervention is reflected in the absolute coordinate system. For this reason, the absolute and machine coordinate systems are the same before and after manual intervention.
The tool path after manual intervention is determined according to the setting of bit 1 (ABS) of parameter No. 7001 as follows.

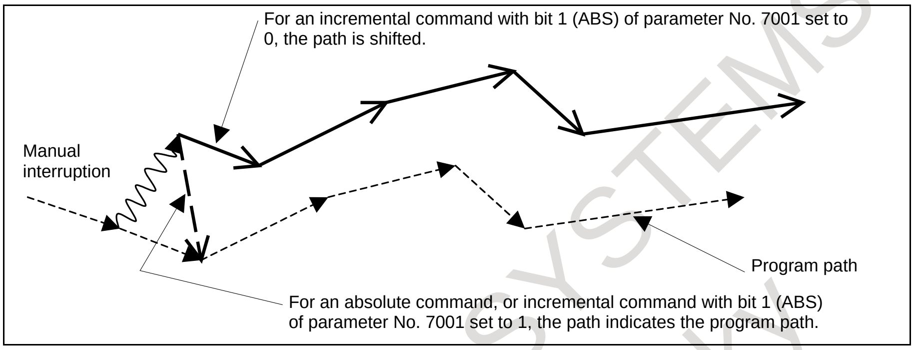


Fig. 5.4 (a)

- When manual absolute turns off

The distance by which the tool is moved is reflected in the absolute position display during manual operation.
When the system is reset or restarted in the automatic operation mode after manual operation, however, the absolute position display indicates the position before manual intervention and the absolute coordinate system after manual intervention is shifted from the machine coordinate system by the amount of manual intervention. The tool path after manual intervention is as follows.

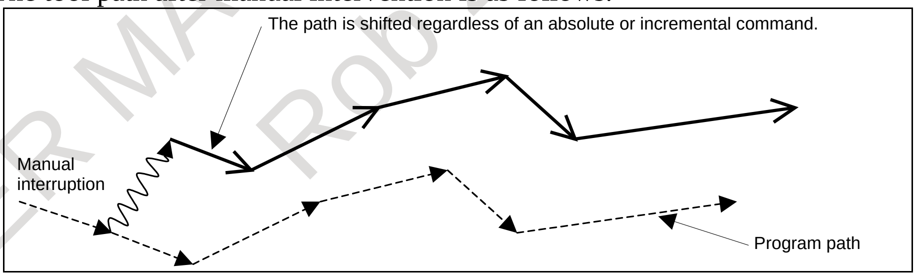


Fig. 5.4 (b)

Signal

Manual absolute signal *ABSM<Gn006.2>

[Classification] Input signal
[Function] Turns the manual absolute function on or off.
[Operation] When this signal is set to 0, turns on the manual absolute function.
When this signal is set to 1, turns off the manual absolute function.

Manual absolute check signal MABSM<Fn004.2>

[Classification] Output signal

[Function] Notifies the state of the manual absolute signal.

[Output cond.] This signal is set to 1 in the following case:

- When the manual absolute signal *ABSM is set to 0

This signal is set to 0 in the following case:

- When manual absolute signal *ABSM is set to 1

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn006						*ABSM		
Fn004						MABSM		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7001							ABS	

[Input type] Parameter input

[Data type] Bit path

#1 ABS For the move command after manual intervention in the manual absolute on state:

0: Different paths are used in the absolute (G90) and incremental (G91) modes.

1: The same path (path in the absolute mode) is used in the absolute (G90) and incremental (G91) modes.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Manual absolute on/off

5.5 OPTIONAL BLOCK SKIP/ADDITION OF OPTIONAL BLOCK SKIP

Overview

When a slash followed by a number (/n, where n = 1 to 9) is specified at the head of a block, and optional block skip signals BDT1 to BDT9 are set to 1 during automatic operation, the information contained in the block for which /n, corresponding to signal BDTn, is specified is ignored (from /n to the end of the block).

(Example)

/2 N123 X100.0 Y200.0 ;

Table 5.5 (a)

Input signal	Code specified at the head of a block
BDT1	/ or /1 (Note)
BDT2	/2
BDT3	/3
BDT4	/4
BDT5	/5
BDT6	/6
BDT7	/7
BDT8	/8
BDT9	/9

NOTE

Number 1 for /1 can be omitted. However, when two or more optional block skip switches are used in one block, number 1 for /1 cannot be omitted.

(Example)

//3 N123 X100.0 Y200.0 ; Invalid

/1 /3 N123 X100.0 Y200.0 ; Valid

The following figures show the relationship between the timing, when optional block skip signals (BDT1 to BDT9) are set to 1, and the ignored information:

1. When BDTn is set to 1 before the CNC starts reading a block containing /n, the block is ignored.

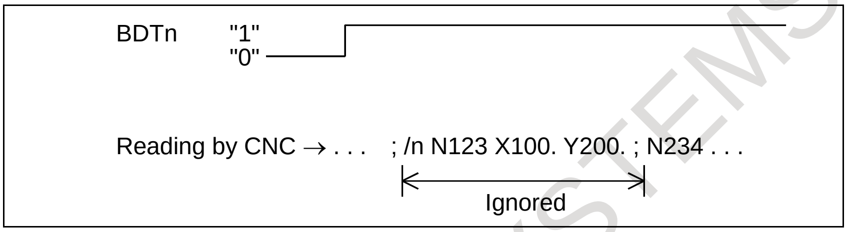


Fig. 5.5 (a)

2. When BDTn is set to 1 while the CNC is reading a block containing /n, the block is not ignored.

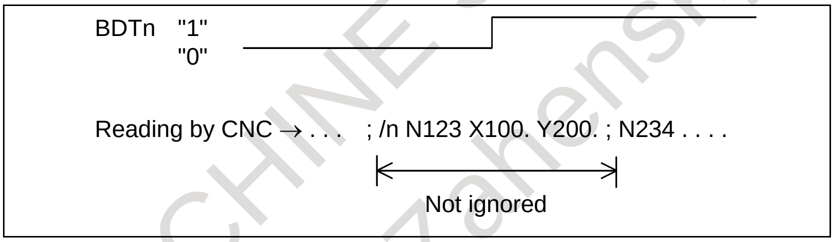


Fig. 5.5 (b)

3. When BDTn, currently set to 1, is set to 0 while the CNC is reading a block containing /n, the block is ignored.

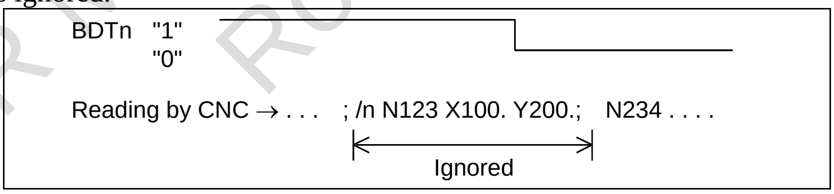


Fig. 5.5 (c)

4. Two or more optional block skip switches can be specified in a block and the block is ignored when BDTn corresponding to one of them is set to 1.

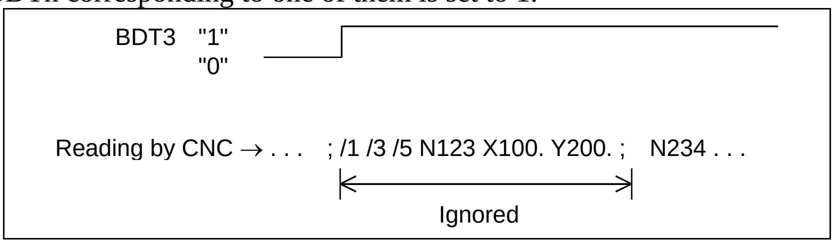


Fig. 5.5 (d)

Note

> **NOTE**
> 1. This function is ignored when programs are loaded into memory. Blocks containing /n are also stored in memory, regardless of how the optional block skip signal is set.
>   Programs stored in memory can be output, regardless of how the optional block skip signals are set.
>   Optional block skip is effective even during sequence number search operation.
> 2. Position of a slash
>   A slash (/) must be specified at the head of a block. If a slash is placed elsewhere, the information from the slash to immediately before the EOB code is ignored.
> 3. TV and TH check
>   When an optional block skip signal is 1. TH and TV checks are made for the skipped portions in the same way as when the optional block skip switch is 0.
> 4. When bit 6 (BDA) of parameter No.11504 is 0 and the optional block skip is invalid and /2 to /9 is specified, the specifying of optional block skip is ignored and the block is executed. When bit 6 (BDA) of parameter No.11504 is 1 and the optional block skip is invalid and /2 to /9 is specified, the alarm PS0004 "INVALID BREAK POINT OF WORDS" is generated.

Signal

Optional block skip signals BDT1<Gn044.0>, BDT2 to BDT9<Gn045>

[Classification] Input signal

[Function] Select whether a block containing /n is to be executed or ignored.

[Operation] During automatic operation, a block containing /n in the program is ignored when the corresponding optional block skip signal is set to 1. It is executed normally when the signal is set to 0.

Optional block skip check signals MBDT1<Fn004.0>, MBDT2 to MBDT9<Fn005>

[Classification] Output signal

[Function] Notify the PMC of the states of the optional block skip signals BDT1 to BDT9. Nine signals are provided, corresponding to the nine optional block skip signals. Signal MBDTn corresponds to signal BDTn.

[Output cond.] Signal MBDTn is set to 1 in the following case:
- When the corresponding optional block skip signal (BDTn) is set to 1

Signal MBDTn is set to 0 in the following case:
- When the corresponding optional block skip signal (BDTn) is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn044								BDT1
Gn045	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2

	#7	#6	#5	#4	#3	#2	#1	#0
Fn004								MBDT1
Fn005	MBDT9	MBDT8	MBDT7	MBDT6	MBDT5	MBDT4	MBDT3	MBDT2

Parameter

Parameters which are used for this function are as follows.

	#7	#6	#5	#4	#3	#2	#1	#0
11504		BDA						

[Input type] Parameter input
[Data type] Bit

#6 BDA When the optional block skip is disabled and /2 to /9 is specified:
0: Alarm is not generated.
1: Alarm PS0004 "INVALID BREAK POINT OF WORDS" is generated.

> **NOTE**
> 1 This parameter is disabled to sequence number search operation.
> 2 This parameter is disabled to macro executor (conversational macro, auxiliary macro).

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Optional block skip/addition optional block skip

5.6 PROGRAM RESTART

Overview

When the tool is broken during automatic operation, or when a machining operation interrupted by a holiday is to be restarted after the holiday, you can restart machining from a desired block by specifying the sequence number of the block or the number of blocks from the beginning of the program to the block where machining is to restart.
This function can also be used as a high-speed program check function.
There are two types of restarting methods as follows.
P type: Restart after a tool is broken down
Q type: Restart after the power is turned off (after a holiday, etc.) or after emergency stop is canceled

Explanation

- Outputting M, S, T, and B (second auxiliary function) codes on the program restart screen

By setting bit 7 (MOU) of parameter No. 7300, the M, S, T, and B (second auxiliary function) codes can be output after a block where the program is to restart is searched for.

1 Before a movement to the machining restart point

<1> The last M, S, T, and B (2nd auxiliary function) codes can be output to the PMC automatically. The last S code is output as the maximum spindle speed if the S code is specified together with G92 in one block; otherwise, the S code is output as a specified spindle speed. On the program restart screen, only the S code specified last is indicated regardless of whether it is specified together with G92 in one block or not.

<2> All M codes sampled during the search for the block where the program is to restart and the last S, T, and B (second auxiliary function) codes can be output to the PMC automatically.
Up to 35 M cods can be sampled. If there are more than 35 M codes to be sampled, the latest 35 M codes are output to the PMC.

<1> or <2> above can be selected by setting bit 6 (MOA) of parameter No. 7300.

2 Until the machining restart point is reached (over store mode)
On the program restart screen, you can specify M, S, T, and B (second auxiliary function) codes through MDI key input while maintaining the MEM mode or DNC mode.

- Movement to the restart point

No.7310 sets the ordinal number of an axis along which a movement is made in dry run after program restart. When No.7310 is not set, alarm SR5020 “PARAMETER OF RESTART ERROR” or the warning “ORDINAL NUMBER ERROR (RESTART)” is displayed.

- Presetting the relative coordinate values after the search

If bit 3 (RPR) of parameter No.7301 is 1, relative coordinate values are preset by absolute coordinate values after the search in program restart.

Limitation

- P type Restart

In the following conditions, P type restart cannot be performed:

- Bit 7 (SQP) of parameter No. 13117 is set to 1.
- Automatic operation has not been performed since power-on.
- Automatic operation has not been performed since emergency stop was released.
- Automatic operation has not been performed since the coordinate system was changed or shifted (change of the external workpiece origin offset value).

The block that can be restored properly by P type program restart is a block for which coordinate system setup or change was performed most recently before machining was interrupted.

- Block number specification Restart

In the following conditions, the block number specification restart cannot be performed:

- Bit 6 (SQB) of parameter No. 13117 is set to 1.
- Specified block number could not be found. (Alarm PS0071 “DATA NOT FOUND” occurs.

- Restart block

The block where the program is to restart is not necessarily be the block at which the program was interrupted. You can restart the program from any block. For P-type restart, however, the block where the program is to restart must use the same coordinate system as when program execution was interrupted.

- Single block

When the single-block operation is enabled at the time of a movement to the restart point, a single-block stop occurs each time an axis operation takes place. In this case, no MDI operation is allowed.

- Manual intervention

During movement to the restart point, manual intervention is allowed for an axis for which a return operation has not yet been performed. However, manual operations do not cause any movement along axes for which a return operation has already been completed.

- MDI

When the search operation has ended, no move command can be specified by MDI before axis movement.

- Reset

Do not perform a reset operation during the time from the start of the search operation of the restart sequence until machining is restarted.
If a reset operation is performed, the restart steps must be performed again from the beginning.

- Feed hold
If a feed hold operation is performed during the search, the restart steps must be performed again from the beginning.

- Manual absolute
Every manual operation must be performed with the manual absolute mode turned on regardless of whether the manual operation is performed before or after machining.

- Reference position return
Unless an absolute position detector (absolute pulse coder) is provided, be sure to perform reference position return after power-up, then perform restart operation.

- Program restart signal
When program restart signal SRN <Gn006.0> is set to "1", pressing the cycle start does not cause a start.

- Blocks specifying a macro statement, macro call, and subprogram call
Blocks specifying a macro statement, macro call, and subprogram call are not searched for even when they have a sequence number. In such a case, search for a block previously preceding such a block.

- Interruption type custom macro
During movement to the machining restart point at a dry run feedrate, no interruption type custom macro can be started. If an interruption type custom macro is started, alarm DS0024 is issued.

- Commands that prevent program restart
Program restart cannot be performed for blocks placed in the following modes:
- Polar coordinate interpolation (G12.1)
- Threading (G32, G33), Circular threading (G35, G36), Threading cycle (G92), Multiple repetitive threading cycle (G76)
- Polygon turning (G50.2)
- Balance cutting (G68)
- Rigid tapping
- Spindle positioning

If any of the following commands is included between the beginning of a program and the block where the program is to restart, program restart cannot be performed:
- Workpiece coordinate system preset (G92.1, G50.3)
- Commands for enabling and disabling synchronous/mixture control and superimposed control
- Commands for enabling and disabling axis synchronous control

- M, S, and T commands not usable in over store mode
The M, S, and T functions listed below, unlike the other M, S, and T functions, have special meanings within the CNC. These M, S, and T commands cannot be specified from the over store screen. To specify these commands, cancel the over store mode, and execute them in MDI operation.
Example:
- Spindle positioning
- Rigid tapping
- Tool management
- Tool offset for lathe system (T code)

- Tool radius / Tool nose radius compensation
When bit 4 (INT) of parameter No. 13117 is set to 1, the interference check on Tool radius/tool nose radius compensation can be disabled during search.

- Cs contour control

Usually, before a program restart operation can be started, it is necessary to enter Cs contour control mode and establish the origin of the Cs contour controlled axis.
By using the Cs axis coordinate establishment function and setting bit 5 (CCS) of parameter No. 7300 to 1, however, it is no longer necessary to enter Cs contour control mode or establish the origin of the Cs contour controlled axis before starting a program restart operation.
In this case, it is possible to change to Cs contour control mode, if necessary, after the end of the search for the block subject to the restart.
If the system enters Cs contour control mode, the origin of the Cs contour controlled axis is established with the Cs axis coordinate establishment function. And, the "DISTANCE TO GO" of the Cs contour controlled axis is recalculated.

> **NOTE**
> To use Cs contour control axis coordinate establishment, perform reference position return for the Cs contour control axis at least once after power on.
> For details, refer to “Cs Contour Control Axis Coordinate Establishment” in this manual.

- Flexible synchronization control

Setting bit 0 (FRS) of parameter No. 13421 to 1 enables program restart in a program containing an M code for turning the flexible synchronization mode on or off.

1. Before starting program restart, set the flexible synchronization control mode selection signal MTA <Gn197.0>, MTB <Gn197.1>, MTC <Gn197.2>, or MTD <Gn197.3> to set the synchronization mode specified in the program command in the block where the program is to restart (the state specified by an M code for turning the flexible synchronization mode on or off).
   If the signal setting is inconsistent with the state specified by the M code, alarm PS5374 occurs. The synchronization mode is assumed to be off in the program from the beginning of the program until the first M code for turning the flexible synchronization mode on or off is specified.
2. The flexible synchronization mode cannot be changed during the execution of program restart (when signal SRNMV <Fn002.4> is set to "1"). If an attempt is made to change the mode, alarm PS5375 occurs.
3. When a block in the flexible synchronization mode is specified as the restart block, 0 is displayed for DISTANCE TO GO for the slave axis. In addition, “********” is displayed as the restart position for the slave axis.
4. To restart a program from a block after the flexible synchronization mode is canceled, an absolute command must be specified for the slave axis after the mode is canceled. If no absolute command is specified for the slave axis, alarm PS5378 occurs.

- 3-dimensional coordinate system conversion / Tilted working plane command

When the block where the program is to restart is in 3-dimensional coordinate system conversion mode G68 (machining center system) or G68.1 (lathe system) or Tilted working plane command, bit 1 (3DD) of parameter No. 7301 can be set to select whether to move the tool to the restart position along each axis according to the program coordinate system or workpiece coordinate system. When the bit (3DD) is set to 0, the tool moves to the restart position along each axis according to the program coordinate system; when it is set to 1, the tool moves according to the workpiece coordinate system.
The “DESTINATION” and “DISTANCE TO GO” values are displayed according to the setting of this bit (3DD).
The "ABSOLUTE" values are displayed according to the setting of bit 6 (DAK) of parameter No. 3106.

- Setting to prevent movement to restart position

When there are two or more axes which has the same setting value in parameter No.7310, the movement to the restart position is not executed in the second or more axis.

Signal

Program restart signal SRN<Gn006.0>

[Classification] Input signal

[Function] Selects program restart.

[Operation] When the program restart signal is set to "1" to search for the sequence number of the block to be restarted, the screen changes to the program restart screen. When the program restart signal is set to "0", and automatic operation is activated, the tool is moved back to the machining restart point at dry run speed along the axes one by one in the sequence specified in parameter No. 7310. When the tool is set to the restart point, machining restarts.

Program restart under way signal SRNMV<Fn002.4>

[Classification] Output signal

[Function] Indicates the program is being restarted.

[Output cond.] The program restart under way signal becomes "1" when:

- The program restart signal is set to "0" after the screen changes to the program restart screen.

The signal is set to "0" when:

- The program restart sequence ends (the tool has been moved to the restart point on all controlled axes).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn006								SRN
Fn002				SRNMV				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7300	MOU	MOA	CCS					

[Input type] Parameter input

[Data type] Bit path

#5 CCS When the Cs contour control axis is used in the spindle mode or when the origin of the Cs contour control axis is not established, program restart is:

0: Disabled.

1: Enabled.

#6 MOA In program restart operation, before movement to a machining restart point:

0: The last M, S, T, and B codes are output.

1: All M codes and the last S, T, and B codes are output.

This parameter is enabled when the bit 7 (MOU) of parameter No.7300 is set to 1.

#7 MOU In program restart operation, before movement to a machining restart point after restart block search:

0: The M, S, T, and B codes are not output.

1: The last M, S, T, and B codes are output.

	#7	#6	#5	#4	#3	#2	#1	#0
7301					RPR		3DD	ROF

[Input type] Parameter input

[Data type] Bit path

#0 ROF When the coordinates for restarting are displayed on the program restart screen:

0: Tool length compensation (M series), tool position compensation (T series), cutter compensation (M series), and tool-nose radius compensation (T series) are considered.

1: Whether these compensation values are considered depends on the settings of bit 6 (DAL) of parameter No. 3104, bit 7 (DAC) of parameter No. 3104, and bit 1 (DAP) of parameter No. 3129 (parameters for specifying whether to consider each compensation value).

#1 3DD In program restart operation, when the restart block is in 3-dimensional coordinate conversion mode G68 (machining center system) or G68.1 (lathe system), the tool moves to the restart point along each axis:

0: According to the program coordinate system in dry run.

1: According to the workpiece coordinate system in dry run.

The restart coordinates and restart travel distance are also displayed in the coordinate system set in this parameter.

> **NOTE**
> The change made to this parameter in program restart operation is ignored.

#3 RPR When the search is completed in program restart or quick program restart, relative coordinate values are:

0: Not preset.

1: Preset by absolute coordinate values.

> **NOTE**
> This parameter is invalid in direct jump method of quick program restart.

7310	Ordinal number of an axis along which a movement is made in dry run after program restart

[Input type] Setting input
[Data type] Byte axis
[Valid data range] 1 to (Number of controlled axes)

This parameter sets the ordinal number of an axis along which a movement is made in dry run after the program is restarted.

> ⚠ **WARNING**
> 1 When there is two or more axes which has the same setting value in this parameter, the movement to the restart point is not executed in the second or more axis.
> 2 When the quick program restart is effective and bit 6 (SAV) of parameter No.11250 is set to 1, the movement to the restart point is not done in the axis which this parameter is set negative value(-1 to -(Number of controlled axes)).

> **NOTE**
> When this parameter is 0, alarm SR5020, "PARAMETER OF RESTART ERROR" or the warning "ORDINAL NUMBER ERROR (RESTART)" is displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
13117	SQP	SQB		INT				

[Input type] Parameter input
[Data type] Bit path

#4 INT During a program restart, the interference check on cutter/tool nose radius compensation is:
0: Enabled.
1: Disabled.

#6 SQB A program restart with a block number specification is:
0: Enabled.
1: Disabled.

#7 SQP A program restart with the P type is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
13421								FRS

[Input type] Parameter input
[Data type] Bit path

NOTE
Set these parameters for the first path only. It will be effective to all paths.

#0 FRS In a program containing an M code for turning the flexible synchronization mode on/off, a program restart is:
0: Disabled.
1: Enabled.

Alarm and message

Number	Message	Description
PS0060	SEQUENCE NUMBER NOT FOUND	In the program restart sequence number specification, the specified sequence number could not be found. Check the restart block.
PS0071	DATA NOT FOUND	In the program restart block number specification, the specified block number could not be found. Check the restart block.
PS0094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to the Operator's Manual.
PS0095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece offset amount changed.) Perform the correct operation according to the Operator's Manual.
PS0096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece offset amount changed.) Perform the correct operation according to the Operator's Manual.

Number	Message	Description
PS0097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON, after emergency stop or alarms PS0094 to 0097 reset, no automatic operation is performed.) Perform automatic operation.
PS0098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search. Perform the reference position return.
PS0099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.
PS5374	FSC MODE MISMATCH IN RESTART	The current flexible synchronization mode differs from the flexible synchronization mode specified in a programmed command in the program restart block. Check the restart block.
PS5375	FSC MODE CAN NOT CHANGED	The flexible synchronization mode was changed during the execution of program restart. Check the restart block.
PS5378	INVALID RESTART BLOCK	The block specified as the restart block after the flexible synchronization mode was canceled was not a block after an absolute command for the axis specified as the slave axis. Check the restart block.
SR5020	PARAMETER OF RESTART ERROR	An invalid value is set in parameter No. 7310, which specifies the axis order in which the tool is moved along axes to the machining restart position in dry run. A value ranging from 1 to the number of controlled axes may be set in this parameter.
DS0024	UINT SIGNAL WAS ILLEGALLY INPUT	An interruption type custom macro was called during movement to the program restart position. Please do not call the interruption type custom macro.

Warning

⚠ **WARNING**

As a rule, the tool cannot be returned to a correct position under the following conditions.

Special care must be taken in the following cases since none of them cause an alarm:

- Manual operation is performed when the manual absolute mode is OFF.
- Manual operation is performed when the machine is locked.
- When the mirror image is used. However, P type return is possible for a block that switched between ON and OFF most recently or a subsequent block. In this case, the mirror image signal status present when the program was interrupted must be maintained.
- When no coordinate system is set up at the beginning of a program in which main commands are executed in the incremental mode.
- When manual intervention is performed in the course of axis movement for returning operation.
- When the program restart is commanded for a block between the block for skip cutting and subsequent absolute command block.
- When program restart is specified in the machine lock state, then the machine lock is canceled.
- When program restart specified for an intermediate block for a multiple repetitive canned cycle
- In general, when a coordinate system is set up, changed, or shifted after the search operation ends, the tool cannot be returned to a correct position.

Caution

> ⚠ **CAUTION**
> 1 Keep the following in mind when restarting a program including macro variables.
> - Common variable
>   When the program is restarted, the previous values are inherited as common variables without being preset automatically. Before restarting the program, initialize the appropriate variables to the original values used at start of the previous automatic operation.
> - System variable
>   The following system variables are not updated during search for restart:
>   #5021 to #5040 and #100051 to #100100 ABSMT
>   #5041 to #5060 and #100101 to #100150 ABSOT
>   #5061 to #5080 and #100151 to #100200 ABSKP
> - DI/DO
>   At restart of the program, DI can be read by a system variable, but DO cannot be output.
> - Clock
>   When the program is being restarted, the clock time can be obtained by a system variable, but the time cannot be preset.
> - Tool offset and workpiece origin offset
>   When the program is being restarted, the offset can be read by a system variable, but change of the offset is allowed only for the Q type.
>
> 2 The M, S, T, and B (second auxiliary function) codes specified in the over store mode are not displayed on the program restart screen.
> 3 In the over store mode, the over store mode is not cancelled even though the operation mode is changed to a mode other than the MEM mode or RMT mode, but a value cannot be input in the over store item.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Program restart

5.6.1 Auxiliary Function Output in Program Restart Function

Overview

This function provides the following features for program restart:

- M/S/T/B codes found during a search through a block to be restarted for operation are output to the program restart screen and an MDI program. Then the M/S/T/B functions can be executed from the MDI program.
- On a system having the M-code grouping feature, M codes are grouped. When they are output to the above MDI program, only the M code specified last, among the M codes in the same group, is output to the program restart screen and MDI program.
- M codes for calling subprograms/custom macros and their arguments are also output to the MDI program.
- The order in which individual axes move the tool to the machining restart position can be not only conventionally parameter-set but also set from the program restart screen.

Explanation

- Program restart procedure

Follow the procedure below to restart a program using this function:

1. Turn on the program restart switch. (Set program restart signal SRN <Gn006.0> to 1.)
2. Display a program you want to restart and search for the beginning of the program.
3. Enter the sequence number (Nxxxx) or block number (Bxxxx) of a block you want to restart, then press the soft key [P TYPE] or [Q TYPE].
4. Upon completion of searching the block, the program restart screen will be displayed.
5. The M/S/T/B codes found by the search are output to the MDI program. At this time, program restart MDI program output completion signal SQMPR <Fn316.6> becomes 1. (For details, see "M/S/T/B code output to the MDI program".)
6. On the program restart screen, set the order in which individual axes moves the tool to the machining restart position.
   (For details, see "Specifying the order in which axes move the tool to the machining restart position".)
7. Turn off the program restart switch. (Set program restart signal SRN <Gn006.0> to 0.) At this time, the number to the left of each axis name under "DISTANCE TO GO" blinks, and the program restart underway signal <Fn002.4> becomes 1.
8. Switch to MDI mode and check the MDI program. Fix the MDI program as required, then execute cycle start. When the execution of the MDI program is started, the program restart MDI program output completion signal SQMPR <Fn316.6> becomes 0. (When program restart signal SRN <Gn006.0> is 0 and program restart MDI program output completion signal SQMPR <Fn316.6> is 1, the PMC ladder can be used to switch to MDI mode and execute cycle start.)
9. When the execution of the MDI program is completed, return to MEM mode. (Upon completion of MDI program execution, program restart MDI program execution completion signal SQMPRE <Fn316.7> becomes 1. When SQMPRE is 1, the PMC ladder can be used to return to MEM mode.)
10. Check whether the distance in "DISTANCE TO GO" is correct and whether the tool does not hit the workpiece or any other object when moving to the machining restart position. If the tool hits anything, move the tool manually to a position free from interference.
11. Press the cycle start button. The tool moves on each axis in the order set in step 6 to the machining restart position at the dry run feedrate, then restarts machining.

> **NOTE**
> 1. In steps 4, 7, and 11, the warning "ORDINAL NUMBER ERROR (RESTART)" is displayed if the order in which individual axes move the tool is set incorrectly.
> 2. If the warning occurs in step 4 or 7, you can perform the step that follows.
> 3. If the warning occurs in step 11, cycle start cannot be executed.
>   In this case, correct the order in which axes move the tool to the program restart position, then perform step 11 again.

- Protecting the MDI program

If you set bit 5 (PMP) of parameter No. 13117 to 1 and memory protect signal KEY3 <G0046.5> or KEY1 <G0046.3> to 0, the output MDI program remains protected until switching to MEM mode after the execution of the MDI program is completed. (That is, the MDI program is protected after the program restart MDI program output completion signal SQMPR <Fn316.6> becomes 1 until the program restart MDI program execution completion signal SQMPRE <Fn316.7> becomes 0.) You cannot edit the MDI program being protected. If you press INSERT, DELETE, or CALC ALTER, the warning "WRITE PROTECT" is displayed.

NOTE

1. When bit 7 (KEY) of parameter No. 3290 is set to 0, KEY3 is used to protect the MDI program. When the bit is set to 1, KEY1 is used instead.
2. When the 8-level data protection function is enabled, whether MDI program editing is allowed or not depends on the setting of that function. (It does not depend on the memory protect key setting.)

Signal

Memory protect signals KEY3 <G0046.5>, KEY1 <G0046.3>

[Classification] Input signal

[Function] Enables MDI program editing.

[Operation] KEY1 or KEY3 enables editing of the MDI program of MSTB code output by the "auxiliary function output in program restart" function.
When bit 7 (KEY) of parameter No. 3290 is 0, KEY3 is enabled. When the bit 1, KEY1 is enabled instead.

Program restart MDI program output completion signal SQMPR <Fn316.6>

[Classification] Output signal

[Function] Reports that M/S/T/B code output to the MDI program for program restart has been completed.

[Operation] 1) This signal becomes 1 when:

- M/S/T/B code output to the MDI program for program restart has been completed.

NOTE

If there is no M/S/T/B code to be output, no MDI program is created. In that case, SQMPR does not become 1.

2) This signal becomes 0 when:

- The execution of the MDI program is started.
- Program restart operation terminates before the execution of the MDI program is started.
- A reset occurs before the execution of the MDI program is started.

Program restart MDI program execution completion signal SQMPE <Fn316.7>

[Classification] Output signal

[Function] Reports that the execution of the MDI program output upon program restart has terminated.

[Output cond.] 1) This signal becomes 1 when:

- The execution of the MDI program output upon program restart has terminated.

2) This signal becomes 0 when:

- The machine switches to MEM mode after the MDI program output upon program restart has been executed.
- A reset occurs after the MDI program output upon program restart has been executed.

NOTE

To use the SQMPR and SQMPRE signals, the "auxiliary function output in program restart" function is required.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0046			KEY3		KEY1			

	#7	#6	#5	#4	#3	#2	#1	#0
Fn316	SQMPE	SQMPR						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3290	KEY							

[Input type] Parameter input
[Data type] Bit path

#7 KEY For memory protection keys:
0: The KEY1, KEY2, KEY3, and KEY4 signals are used.
1: Only the KEY1 signal is used.

> **NOTE**
> 1 The functions of the signals depend on whether KEY=0 or KEY=1.
> When KEY = 0:
> - KEY1: Enables a tool offset value, workpiece zero point offset value, and workpiece shift value to be input.
> - KEY2: Enables setting data, macro variables, and tool life management value to be input.
> - KEY3: Enables program registration and editing.
> - KEY4: Enables PMC data (counter and data table) to be input.
>
> When KEY = 1:
> - KEY1: Enables program registration and editing, and enables PMC parameter input.
> - KEY2 to KEY4: Not used
>
> 2 When a multi-path system is used, the setting for path 1 is followed.

	#7	#6	#5	#4	#3	#2	#1	#0
13117			PMP					

[Input type] Parameter input
[Data type] Bit path

#5 PMP To the MDI program that is output due to a program restart, the memory protection signals KEY1 and KEY3 are:
0: Not effective.
1: Effective.

> **NOTE**
> When 0 is set in bit 7 (KEY) of parameter No. 3290, KEY3 is used to protect the MDI program. When 1 is set in the parameter, KEY1 is used.

Notes

To use this function, be sure to set bit 7 (MOU) of parameter No. 7300 to 0.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	"Auxiliary function output in program restart" function

5.6.2 Approach for Each Arbitrary Axis in Program Restart

Overview

For the approach to the program restart position in program restart, the following functions are implemented. These functions allow the operator to sequentially determine the order in which the tool moves along axes with operations on the screen according to the situation including obstacles.

- On the program restart screen, enter the axis name of an arbitrary axis and press soft key [MOVE AXIS] to set an axis along which to move the tool to the program restart position. Then, execute cycle start. The tool moves to the program restart position along the target axis at the dry run feedrate. After the tool moves to the program restart position along all axes, executing cycle start restarts machining.
- When the tool has moved to the program restart position along not all of axes (including the case where the tool does not move along any axis), executing cycle start moves the tool to the program restart position at the dry run feedrate. At this time, the tool moves sequentially along the axes in the order specified in parameter No. 7310. Machining is then restarted.

Parameter

7310	Ordinal number of an axis along which a movement is made in dry run after program restart

[Input type] Setting input
[Data type] Byte axis
[Valid data range] 1 to (Number of controlled axes)

This parameter sets the ordinal number of an axis along which a movement to the restart point is made in dry run after the program is restarted.

> ⚠ **WARNING**
> 1 When there is two or more axes which has the same setting value in this parameter, the movement to the restart point is not executed in the second or more axis.
> 2 When the quick program restart is effective and bit 6 (SAV) of parameter No.11250 is set 1, the movement to the restart point is not done in the axis which this parameter is set negative value(-1 to -(Number of controlled axes)).

> **NOTE**
> When this parameter is 0, alarm SR5020, "PARAMETER OF RESTART ERROR" or the warning "ORDINAL NUMBER ERROR (RESTART)" is displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
11250	OAA							

[Input type] Parameter input
[Data type] Bit path

 # 7 OAA In the program restart function, the approach to the program restart position for each arbitrary axis is:
0: Not used.

1: Used.

Warning and message

Warning message	Description
THERE IS NO SPECIFIED NAME OF AXIS	An invalid axis was specified as an input value. Enter a correct axis name.
WRONG MODE	The CNC mode is invalid. In the MEM or DNC mode, set or cancel an axis.
COMMAND ILLEGAL USE	1 The program restart coordinates are not set. 2 The tool starts moving to the program restart position along another axis. 3 No axis is set. (only for cancellation)

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Program Restart

5.7 QUICK PROGRAM RESTART

Overview

Generally, the following operations are required when machining is interrupted due to a reason such as a damaged tool and is restarted:

1. Removes the cause which interrupted the machining.
2. Locates the interrupted point in the program.
3. Restores the machine to the suitable status (including the auxiliary function status and spindle status).
4. Moves the tool to the position suitable for restarting the machining.
5. Resumes automatic operation from the interrupted block or before several blocks.

This function helps to perform the above operations from 2 to 5 more easily.

- For this function, a screen for restart operation is available. This screen is called “program restart setting screen”. To perform program restart, first specify a target block on the program restart setting screen, next press soft key [SEARCH EXEC]. Then, program restart is executed.
- The program restart setting screen is also used to locate the interrupted point in the program. On the program restart setting screen, information (including the program name and sequence number) on the last executed block is displayed as primary information. This information can be checked to locate the interrupted point in the program.
- On the program restart setting screen, in addition to the information on the last executed block, information on some blocks executed before that can be displayed. This information is automatically stored in CNC memory during the operation of a program. This memory is called “program restart memory”.
  Block information stored in program restart memory is called “restart block information”.
- You can select an arbitrary block in restart block information and press soft key [SEARCH EXEC] to restart automatic operation from that block.

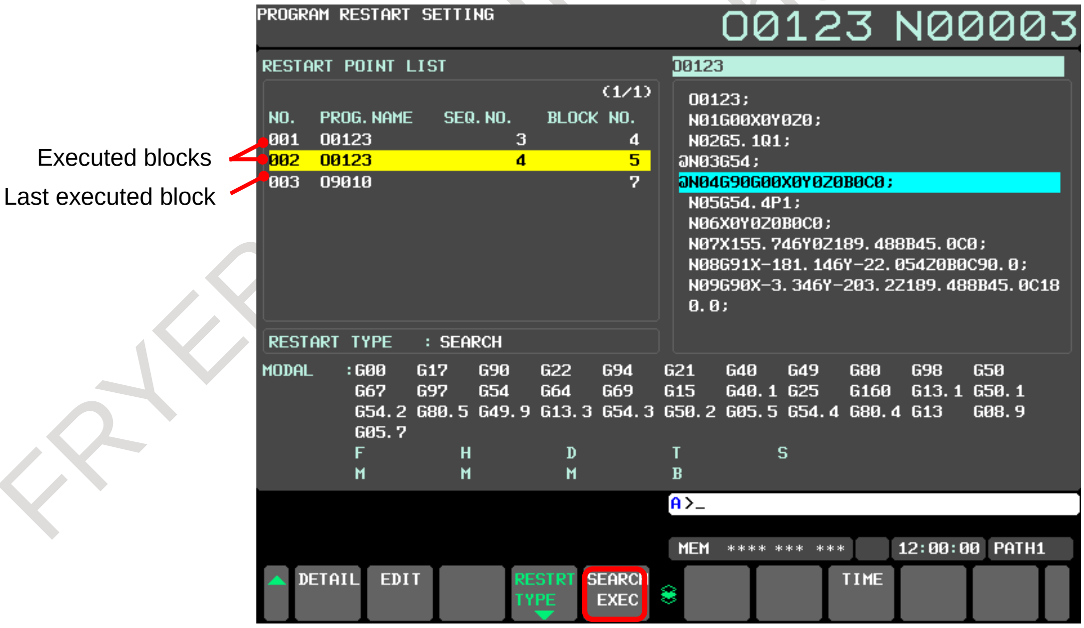


Fig. 5.7 (a) Program restart setting screen

- By changing a displayed block number or sequence number using the MDI unit and pressing [SEARCH EXEC] on the program restart setting screen, the program is restarted according to the data after change. With this operation, the program can be restarted from a position shifted relatively from the block stored in the program restart memory.
- To restore the machine to the suitable status (including the auxiliary function status and spindle status), use the “auxiliary function output in program restart” function. By using this function, any

auxiliary function found during search is output to the MDI program. This MDI program can be executed to restore the machine status easily.

- For this function, either of the following two types of restart methods can be selected.

  <1> Search method

  Use this type when it is difficult to restore the status of modal information and auxiliary functions manually. This type can be selected to restart a program not only from the block stored in the program restart memory, but also from another block. When this method is selected, the modal information and position information of the block from which to restart the program can be restored automatically.

  Application cases : When the power is turned off after machining is interrupted
  When another workpiece is machined after machining is interrupted, and the restart block information on the previous workpiece is not stored
  When the program is to be restarted from a block which is not stored in the program restart memory

  Function : By using the “auxiliary function output in program restart” function, the auxiliary function status can be restored.
  The tool automatically moves to the restart position along each axis when the program is restarted.

  Operation : Restores modal information and coordinate values while performing the simulated execution of the program until the block specified in the program restart setting screen is found. (Updates modal information and coordinate values without moving the tool along any axis.)
  Mainly updates the following data:
  - Modal information including G, F, D, and H codes
  - Absolute coordinate values
  - Auxiliary function output in the program restart

  <2> Direct jump method

  Use this type when the operator restores the status of modal information and auxiliary functions manually and restarts the machining. This method can be selected only when the program is to be restarted from the block stored in the program restart memory.
  When this method is selected, program restart is performed at high speed, though the modal information and position information of the restart block are not restored. Restart operation will be completed in a short time compared with the search method, in particular, when the restart block is in the middle of a long program.

  Application cases : When the operator can easily restore the machine status because changes in the machine status are few when, for example, machining is interrupted, and is to be restarted immediately)
  When a program is to be restarted from a block where operation can be started without restoring the machine status, such as a block on a machining process boundary

  Function : The “auxiliary function output in program restart” function cannot be used.
  The operator must determine whether to specify required auxiliary functions and specify them.
  The machine must be positioned at the restart position manually before operation is restarted because the tool does not move to the restart position along any axis when the program is restarted.
  Modal G codes and other modal information must be set to the suitable status by MDI operation before operation is restarted.

  Operation : Moves only the program cursor to the specified block at high speed without restoring modal information or coordinate values.

- Executing cycle start after the block specified in the program restart setting screen is found restarts operation from that block. When the search method is selected, the auxiliary function status is restored by the “auxiliary function output in program restart” function and the tool is moved to the restart position before operation is restarted.

Notes

- Program restart memory

1. The following blocks are stored in the program restart memory:
   a. Last executed block
   b. Blocks of positioning (When some blocks are consecutive, the first block is stored)
   c. Blocks of subprogram calls (including M98, M198, G65, G, and M code macro calls and subprogram calls)
   d. Other blocks judged to be suitable for restarting a program (blocks for turning each mode on or off and others that function as a breakpoint in a program)
2. If the program restart memory becomes full, block information is deleted from the older one and the information on a new block is overwritten.
3. The program restart memory is cleared when operation is started in the MEM or RMT mode after a reset.
   The memory is also cleared when the program is changed and restart operation using the search method starts.
4. When a program is restarted with specifying a new restart point, the information on the blocks stored before restart operation is not cleared. After the program is restarted, block information after restart is added following the block information stored before restart.

- Search method

1. Any sequence number in the programmable parameter input (G10L52 or G10L50) mode is not searched for when a search is executed with specifying a sequence number.
2. When a short cut is specified for a rotary axis, the tool also uses a short cut to the machining restart position.
3. If the single-block operation is enabled when the tool moves to the restart position, a single-block stop occurs each time the tool moves along an axis. During this operation, however, no MDI operation is allowed.
4. While the tool is moving to the restart position, manual intervention is allowed for an axis along which the tool does not move to the restart position. The tool does not move manually along any axis along which it has moved to the restart position, however.
5. During movement to the machining restart position at a dry run feedrate, no interruption type custom macro can be started. If an attempt is made to start an interruption type custom macro, alarm DS0024 "UINT SIGNAL WAS ILLEGALLY INPUT" occurs.

- Direct jump method

There are the following limitations.

1. A direct jump may not be able to be selected depending on the block. (Example: Block in the cutter or tool nose radius compensation mode)
   If an attempt is made to select “JUMP” as the restart type for such a block, the warning message, “ILLEGAL DATA”, is displayed and the setting is not accepted.
2. A direct jump can be selected to restart a program from a block stored in the program restart memory. It cannot be used to restart a program from a block which is not stored in the program restart memory. If an attempt is made to use a direct jump for the above block, the warning message, “ILLEGAL DATA”, is displayed.
3. When a direct jump is selected, the status of items including position information, modal information, and macro variables is not restored. The “auxiliary function output in program restart” function is not also executed. For this reason, the operator must restore the status manually with considering the status of the restart block.
4. In the following cases, a direct jump can be selected for a block in a subprogram or external subprogram call:
   a. Subprogram call using M98
   b. Subprogram call using G65
   c. Block in a subprogram on the Data Server operating by M198
   d. Block in a program on the Data Server operating in the RMT mode

In other cases, a direct jump cannot be selected for a block in a subprogram or external subprogram call.
That is, a direct jump cannot be selected for the following blocks:

e. Block in a subprogram specified by an M or G code or a program called by a macro call
f. Block operating in the RMT mode (other than the Data Server)
g. Block in an execution macro

5. Note that there are the following limitations for blocks described in c. and d. above:
   (1) It is not possible to jump to the right position when there is a branch instruction of the custom macro.
   (2) It may take 1 to a few minutes to move the cursor to the restart block in a long program longer than 20 MB.
6. The jump position is shifted if the number of lines in the program is changed by editing the program after interrupted.
7. When a direct jump is selected, the auxiliary function status is not restored by the “auxiliary function output in program restart” function. Restore the status of auxiliary functions by MDI operation before starting subsequent machining if necessary.
8. When a direct jump is selected, modal information is not restored and the tool does not move to the restart position. The status must be restored by MDI operation or manual operation before starting subsequent machining.

- Stop at the restart position

Change single block operation to stop at the restart position. Setting the bit 1 (RPS) of parameter No.7331 to 1 enables to stop at the restart position even for continuous operation.

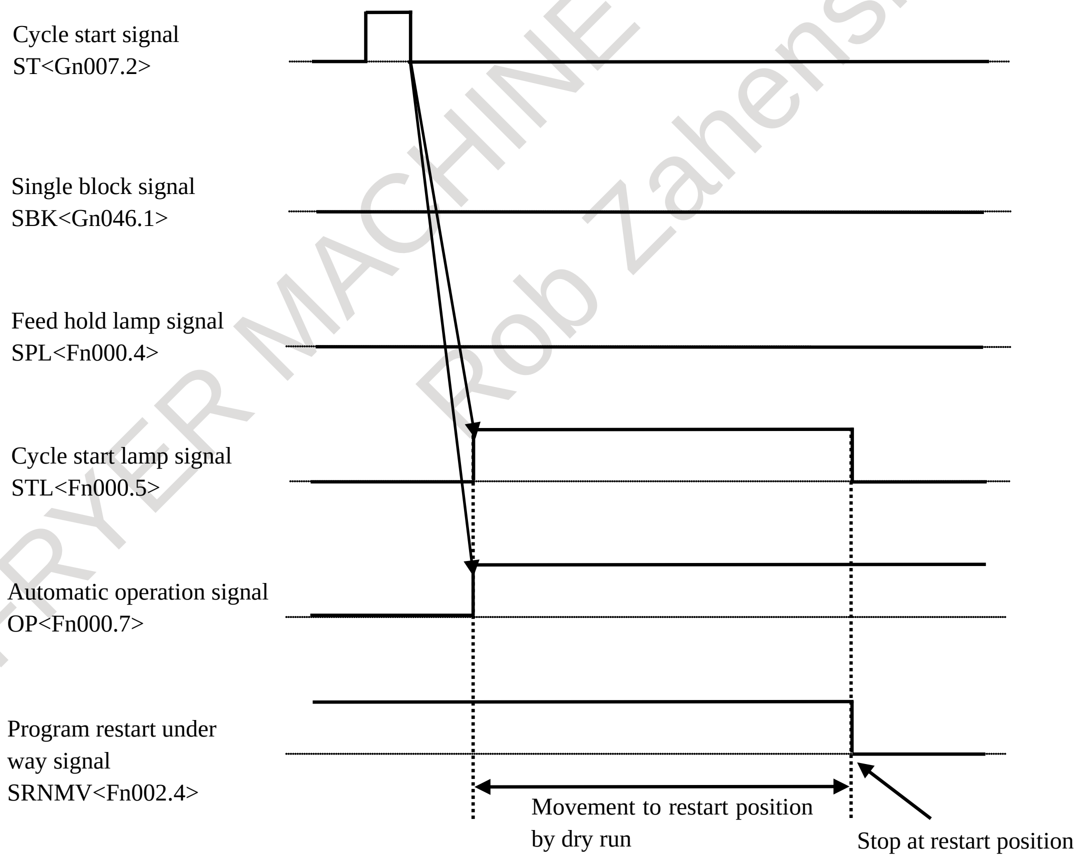


Fig. 5.7 (b) Operation in setting bit 1 (RPS) of parameter No.7331 to 1

- Execution macro (macro executor)

A program cannot be restarted from a block in an execution macro.

- Modal display on the program restart setting screen
The modal display on the program restart setting screen indicates the status before the execution of the stored block. That is, data specified in the stored block is not displayed.

- Reset
Do not perform reset operation from the start of a search for restart to the restart of machining.
If reset operation is performed, perform restart operation from the beginning again.

- Feed hold
If performing feed hold operation during search, perform restart operation from the beginning again.

- Manual absolute
Perform every manual operation with the manual absolute mode turned on regardless of whether the manual operation is performed before or after machining.

- Reference position return
When no absolute position detector (absolute pulse coder) is provided, be sure to perform reference position return after power-on, then perform restart operation.

- System variable
With a custom macro made by a machine tool builder, different operations between normal operation and program restart operation may be required. For this case, the operator can check the running state of program restart with system variable #3008.

#3008 = 0 : Normal operation
#3008 ≠ 0 : Program restart operation

- Relation to other functions
The symbols in the table are described below:

A : Can restart a program from a block in the mode of each function.
NA : Cannot restart a program from a block in the mode of each function. Also cannot restart a program when any of these commands is specified in the first block of the program to the restart block.
*1 : There are some limitations. See Remarks.
*2 : Can restart a program after the status of items including modal information, macro variables, and auxiliary functions is restored by MDI operation or manual operation.
*3 : May restart a program using the “auxiliary function output in program restart” function in some cases.
*4 : Cannot restart a program from a block in the mode of each function.
Can restart a program from a block in which the mode is off when the program contains a block for turning the mode on or off.
*5 : Coordinate values are calculated on the assumption that the block is executed to the end point. Therefore, if the program is restarted from a block without specifying an absolute command at least once following this command, the positioned coordinate values may be different from those actually executed.

Function name	Search method	Direct jump method	Remarks
Automatic operation (Memory operation)	A	A	
DNC operation	A	*1	Only the DNC operation of the Data Server is available on the direct jump method. When the program restart is executed excluding the Data Server, the warning of "ILLEGAL DATA" is displayed and the restart is not started.

Function name	Search method	Direct jump method	Remarks
DNC operation with memory card	A	NA	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Schedule function	NA	NA	In the search method, alarm SR1968 is generated. In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Optional block skip	*1	*1	The optional block skip signal cannot be changed while executing the program restart. The program restart cannot be correctly executed, and restarts in a wrong block when the signal is changed.
Positioning	A	A	
Single direction positioning (G60)	A	A	
Linear interpolation	A	A	
Circular interpolation	A	A	
Helical interpolation	A	A	
Dwell	A	A	
Thread cutting, synchronous cutting	*4	*4	The thread shifts when restarting on the way of the threading blocks continuously commanded.
Multi threading	*4	*4	Same as the above-mentioned
Continuous threading	*4	*4	Same as the above-mentioned
Variable lead thread cutting	*4	*4	Same as the above-mentioned
Circular thread cutting	*4	*4	Same as the above-mentioned
Skip	*5	*2	
High-speed skip	*5	*2	
Multi-step skip	*5	*2	
Continuous high-speed skip	*5	*2	
Torque limit skip	*5	*2	
Polar coordinate interpolation	*4	*4	In the search method, tool position might shift by operating after the program restarts. In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Cylindrical interpolation	*4	*4	Alarm PS176 is generated and restart is not available in both search methods and the direct jump methods.
Polar coordinate command	A	*4	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Normal direction control	A	*4	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Mirror image	NA	*2	In the search method, tool position might shift by operating after the program restarts.
Programmable mirror image	A	*4	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Mirror image for double turret	A	*4	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Figure copying	A	*4	In the direct jump method, the tool path might shift by operating after the program restarts.

Function name	Search method	Direct jump method	Remarks
Coordinate system rotation	A	*4	In the direct jump method, the tool path might shift by operating after the program restarts.
3-dimensional coordinate system conversion	A	*2	If G68(G68.1) is commanded with MDI operation and 3-dimensional coordinate system conversion mode is restored, the direct jump method is available.
Tilted working plane indexing	A	*2	See item “Tilted working plane indexing” for details when a direct jump is selected. See item “Workpiece setting error compensation (or tilted working plane indexing + tool center point control)” to use tool center point control simultaneously.
Workpiece coordinate system	A	*2	
Workpiece coordinate system preset (G92.1/G50.3)	NA	*2	In the search method, tool position might shift by operating after the program restarts.
Coordinate system setting (G92/G50)	A	*2	
Reference position return	A	A	
Reference position return check	A	A	
2nd reference position return	A	A	
3rd/4th reference position return	A	A	
Tool length offset	A	*2	
Tool offset (G45 to G48)	A	*2	
Tool functions (T code on lathe system)	A	*2	
Tool radius / Tool nose radius compensation	A	*4	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Tool offset for Milling and Turning function	A	*2	
Automatic tool length measurement (M series) / Automatic tool offset (T series)	*5	*2	
Plane selection	A	*2	
Balanced cutting	*4	*4	Synchronization between paths might shift by operating after the program restarts. In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Programmable data input	A	A	
Programmable parameter input	A	A	For the blocks between G10L52 and G11, program restart with specifying a sequence number is disabled.
Sub program call (M98)	A	A	
Sub program call (M198)	A	*1	Only a program call on the Data Server is available for a direct jump.
Inch/metric conversion	A	*2	
Direct drawing dimension programming	A	*4	
Chamfering/Corner R	A	*4	
Optional chamfering corner R	A	*4	
Custom macro	A	*2	
Interruption type custom macro	A	*2	
Canned cycle	*4	*4	Tool path might shift by operating after the program restarts.
Multiple repetitive canned cycles	*4	*4	Tool path might shift by operating after the program restarts.

Function name	Search method	Direct jump method	Remarks
Multiple repetitive canned cycles II	*4	*4	Tool path might shift by operating after the program restarts.
Canned cycles for grinding	*4,*5	*4	Tool path might shift by operating after the program restarts.
Canned cycle for drilling (except rigid tapping)	A	*4	In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Small-hole peck drilling cycle	*4	*4	In the search method, operation is restarted from the beginning of the hole machining. In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
Rigid tapping	*4	*4	In the search method, an alarm PS0200 is generated after restart. In the direct jump method, the warning of "ILLEGAL DATA" is displayed and the restart is not started.
PMC axis control	*3	*2	
Real time custom macro	NA	*4	The real time custom macro is not executed while restarting. Therefore, the real time custom macro might not operate correctly after the program restarts. (both search methods and the direct jump methods)
Polygon turning	*4	*4	
Polygon turning by Cs contour control	*4	*4	
Polygon machining with two spindles	*4	*4	
High-speed HRV3 control (G5.4)	*4	*2	On/off of this function is not restored as for the search method either. If G05.4Q1 or Q0 is commanded by MDI operation and the state is restored after restarted, operation can be restarted. (both search methods and the direct jump methods)
Optimum torque acceleration/deceleration	A	A	
Jerk control	A	A	
AI contour control I	A	A	
AI contour control II	A	A	
Exact stop mode	A	*2	
Tapping mode	A	*2	
Cutting mode	A	*2	
Automatic corner override	A	*2	
Auxiliary function	*3	*2	
2nd auxiliary function	*3	*2	
M code group check	*3	*2	
Tool management function	*3	*2	When the search method is selected and a tool exchange macro is executed after search, the amount of tool offset may be incorrect. (For a macro using a command such as H#8409D#8410)
Tool life management	*3	A	
Embedded macro	A	A	
Arbitrary angular axis control	A	A	

Function name	Search method	Direct jump method	Remarks
Axis synchronous control	NA	*4	In the search method, program restart can be restarted if the synchronization is always ON. If on/off of synchronization is changed by M code etc. in the program, the position of the master axis and the slave axis might shift after restart.
Synchronous/Composite control	NA	*4	If on/off is changed by M code etc. in the program, the position of the master axis and the slave axis might shift after restart.
Superimposed Control	NA	*4	If on/off is changed M code etc. in the program, the position of the master axis and the slave axis might shift after restart.
Synchronous/Composite/Superimposed control by program command	NA	*4	If on/off is changed in the program, the position of the master axis and the slave axis might shift after restart.
Flexible synchronization control	*1	*1	See item “Flexible synchronization control”.
Hobbing/Electric gear box	NA	*4	The machining is executed with the function is invalid after restarted.
Electric gear box 2 pair	NA	*4	Same as the above-mentioned
Electric gear box automatic phase synchronization	NA	*4	Same as the above-mentioned
Spindle electric gear box	NA	*4	Same as the above-mentioned
Machining condition selecting function	A	*2	
Spindle positioning	*4	*4	The spindle must be positioned to the restart position by MDI operation in advance.
Cs contouring control	*3	*2	See item “Cs contour control” for details when the search method is selected.
Spindle speed fluctuation detection	A	*2	
Spindle synchronous control	*3	*2	
Live tool control with servo motor	*3	*2	
Servo/spindle synchronous control	*3	*2	
Macro executor	*4	*4	A program cannot be restarted from a block in an execution macro.

- Tool radius / Tool nose radius compensation

When bit 4 (INT) of parameter No. 13117 is set to 1, the interference check on cutter/tool nose radius compensation can be disabled during search by search method.

- Cs contour control

For details, see Item, “Cs contour controlled” in Section, “PROGRAM RESTART”.

- Flexible synchronization control

Setting bit 0 (FRS) of parameter No. 13421 to 1 enables program restart by search method in a program containing an M code for turning the flexible synchronization mode on or off.

1. Before starting program restart, set the flexible synchronization control mode selection signal MTA <Gn197.0>, MTB <Gn197.1>, MTC <Gn197.2>, or MTD <Gn197.3> to set the synchronization mode specified in the program command in the block where the program is to restart (the state specified by an M code for turning the flexible synchronization mode on or off). If the signal setting is inconsistent with the state specified by the M code, alarm PS5374 occurs. The synchronization mode is assumed to be off in the program from the beginning of the program until the first M code for turning the flexible synchronization mode on or off is specified.
2. When program restart is executed (when signal SRNMV <Fn002.4> is set to “1”), the flexible synchronization mode cannot be changed. If an attempt is made to change the mode, alarm PS5375 occurs.

3. When a block in the flexible synchronization mode is specified as the restart block, the restart travel distance for the slave axis is set to 0. In addition, "********" is displayed as the restart position for the slave axis.
4. To restart a program from a block after the flexible synchronization mode is canceled, an absolute command must be specified for the slave axis after the mode is canceled. If no absolute command is specified for the slave axis, alarm PS5378 occurs.

- 3-dimensional coordinate system conversion / Tilted working plane command

When the block where the program is to restart is in 3-dimensional coordinate system conversion mode G68 (machining center system) or G68.1 (lathe system) or Tilted working plane command, bit 1 (3DD) of parameter No. 7301 can be set to select whether to move the tool to the restart position along each axis according to the program coordinate system or workpiece coordinate system. When the bit (3DD) is set to 0, the tool moves to the restart position along each axis according to the program coordinate system; when it is set to 1, the tool moves according to the workpiece coordinate system.
The "DESTINATION" and "DISTANCE TO GO" values are displayed according to the setting of this bit (3DD).
The "ABSOLUTE" values are displayed according to the setting of bit 6 (DAK) of parameter No. 3106.

- Tilted working plane indexing

To restart a program from a block in the tilted working plane indexing mode using a direct jump, follow the procedure below:

1. Return the tool to the restart position.
2. Execute a direct jump.
3. Specify the tilted working plane indexing mode using the MDI unit after the program has been restarted (blinking RSTR disappears).
4. If the program is restarted from block after tool axis direction control, please command tool axis direction control.
5. The present absolute coordinate is matched to the absolute coordinates of the restart block.

- Presetting the relative coordinate values after the search

If bit 3 (RPR) of parameter No.7301 is 1, relative coordinate values are preset by absolute coordinate values after the search in quick program restart. However, this preset function is invalid in direct jump method of quick program restart.

Signal

Program restart under way signal SRNMV<Fn002.4>

[Classification] Output signal
[Function] Indicates the program is being restarted.
[Output cond.] The program restart under way signal becomes "1" when:
- The program restart signal is set to "0" after the screen changes to the program restart screen.

The signal is set to "0" when:
- The program restart sequence ends (the tool has been moved to the restart point on all controlled axes).

Quick program restart under way signal SRNEX<Fn534.1>

[Classification] Output signal
[Function] Indicates the quick program restart is executing.
[Output cond.] The program restart under way signal becomes "1" when:
- Press the soft key [SEARCH EXEC], and quick program restart is begun.

The signal is set to "0" when:
- The quick program restart sequence ends (the tool has been moved to the restart point on all controlled axes).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn002				SRNMV				
Fn534							SRNEX	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3106		DAK						

[Input type] Setting input
[Data type] Bit

#6 DAK Specifies whether to display coordinates in the program coordinate system or workpiece coordinate system as absolute coordinates when the 3-dimensional coordinate conversion mode or the tilted working plane indexing mode is set.
0: Display coordinates in the program coordinate system.
1: Display coordinates in the workpiece coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
3712						CSF		

[Input type] Parameter input
[Data type] Bit

#2 CSF In the Cs contour control mode, the function for setting machine coordinates and absolute coordinates based on the machine position of the spindle if the origin is already set up is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
4353			CSP					

[Input type] Parameter input
[Data type] Bit

#5 CSP The function for transferring Cs axis positional data is:
0: Disabled.
1: Enabled.
When this parameter is used, set the bit 7 (RFCHK3) of parameter No.4016 is to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
7300			CCS					

[Input type] Parameter input
[Data type] Bit path

#5 CCS When the Cs contour control axis is used in the spindle mode or when the origin of the Cs contour control axis is not established, program restart is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
7301					RPR		3DD	ROF

[Input type] Parameter input
[Data type] Bit path

#0 ROF When the coordinates for restarting are displayed on the program restart screen:
- 0: Tool length compensation (M series), tool position compensation (T series), cutter compensation (M series), and tool-nose radius compensation (T series) are considered.
- 1: Whether these compensation values are considered depends on the settings of bit 6 (DAL) of parameter No. 3104, bit 7 (DAC) of parameter No. 3104, and bit 1 (DAP) of parameter No. 3129 (parameters for specifying whether to consider each compensation value).

#1 3DD In program restart operation, when the restart block is in 3-dimensional coordinate conversion mode G68 (machining center system) or G68.1 (lathe system), the tool moves to the restart point along each axis:
- 0: According to the program coordinate system in dry run.
- 1: According to the workpiece coordinate system in dry run.

The restart coordinates and restart travel distance are also displayed in the coordinate system set in this parameter.

> **NOTE**
> The change made to this parameter in program restart operation is ignored.

#3 RPR When the search is completed in program restart or quick program restart, relative coordinate values are:
- 0: Not preset.
- 1: Preset by absolute coordinate values.

> **NOTE**
> This parameter is invalid in direct jump method of quick program restart.

7310	Ordinal number of an axis along which a movement is made in dry run after program restart

[Input type] Setting input
[Data type] Byte axis
[Valid data range] 1 to (Number of controlled axes)

This parameter sets the ordinal number of an axis along which a movement to the restart point is made in dry run after the program is restarted.

> ⚠ **WARNING**
> 1 When there is two or more axes which has the same setting value in this parameter, the movement to the restart point is not executed in the second or more axis.
> 2 When the quick program restart is effective and bit 6 (SAV) of parameter No.11250 is set to 1, the movement to the restart point is not done in the axis which this parameter is set negative value(-1 to -(Number of controlled axes)).

> **NOTE**
> When this parameter is 0, alarm SR5020, "PARAMETER OF RESTART ERROR" or the warning "ORDINAL NUMBER ERROR (RESTART)" is displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
7331							RPS	

[Input type] Setting input
[Data type] Bit

#1 RPS In moving to the restart point of quick program restart, when the single block operation is invalid, the axes
0: do not stop by the restart point.
1: stop in state of the single block by the restart point.

> **NOTE**
> When this parameter is set to 1, the axes stop once in state of the single block in starting after restart operation even if these don't move to the restart point.

	#7	#6	#5	#4	#3	#2	#1	#0
11250	OAA							

[Input type] Parameter input
[Data type] Bit path

#7 OAA In the program restart function, the approach to the program restart position for each arbitrary axis is:
0: Not used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
11251				NPN				

[Input type] Parameter input
[Data type] Bit

#4 NPN In Quick Program Restart Function, when an interrupted program is restarted from the block in the subprogram, the subprogram name is :
0: specified.
1: not specified.

	#7	#6	#5	#4	#3	#2	#1	#0
13117				INT				

[Input type] Parameter input
[Data type] Bit path

#4 INT During a program restart, the interference check on cutter/tool nose radius compensation is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
13421								FRS

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> Set these parameters for the first path only. It will be effective to all paths.

#0 FRS In a program containing an M code for turning the flexible synchronization mode on/off, a program restart is:
0: Disabled.
1: Enabled.

Alarm and message

Number	Message	Description
PS0060	SEQUENCE NUMBER NOT FOUND	In the program restart sequence number specification, the specified sequence number could not be found. Check the restart block.
PS0071	DATA NOT FOUND	In the program restart block number specification, the specified block number could not be found. Check the restart block.
PS0098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search. Perform the reference position return.
PS5374	FSC MODE MISMATCH IN RESTART	The current flexible synchronization mode differs from the flexible synchronization mode specified in a programmed command in the program restart block. Check the restart block.
PS5375	FSC MODE CAN NOT CHANGED	The flexible synchronization mode was changed during the execution of program restart. Check the restart block.
PS5378	INVALID RESTART BLOCK	The block specified as the restart block after the flexible synchronization mode was canceled was not a block after an absolute command for the axis specified as the slave axis. Check the restart block.
DS0024	UINT SIGNAL WAS ILLEGALLY INPUT	An interruption type custom macro was called during movement to the program restart position. Please do not call the interruption type custom macro.

Warning

⚠ WARNING

1 In principle, the tool cannot return to a correct position in the following cases. Special care must be taken in the following cases since none of them cause an alarm.
   - Manual operation is performed when the manual absolute mode is off.
   - Manual operation is performed when the machine is locked.
   - The mirror image is used. The tool can be returned by P type program restart, however, when the program is restarted from a block after the mode is turned on or off last. At this time, keep the state of the mirror image signal when the machining is interrupted.
   - A program in which main commands are executed in the incremental mode does not begin with coordinate system setting.
   - Manual intervention is performed while the tool is returning to the restart position along an axis.
   - Program restart is specified for a block between the block for skip cutting and subsequent absolute command block.
   - After program restart is specified in the machine lock state, the machine lock state is canceled.
   - In general, if a coordinate system is set up, changed, or shifted after search, the tool cannot return to the correct position.

2 When macro variable #3008 (indicating the status of program restart) is used and the behavior of the program during restart operation is different from that during normal operation, the program cannot be restarted from the correct position using the search method.

Example)

Program O0002 performs different operations between normal operation and program restart operation. In the following examples, this program cannot be restarted from the block of N998 or N999 using the search method. A main program including such a subprogram can be restarted, however. In this example, the program can be restarted from the block of N1.

```
O0001(MAIN)             O0002(SUB)
M98P0002                IF[#3008NE0]GOTO999
N1G00X100.Y100.         G53Z0
  :                     T#500
                        N998G00Z0
                        N999
                        M99
```

Caution

> ⚠ **CAUTION**
> Keep the following in mind when restarting a program including a macro variable.
> - Common variable
>   When a program is restarted, the previous values are inherited as common variables without being preset automatically. For this reason, before restarting the program, initialize the required variables to the original values used at the start of the previous automatic operation.
> - System variable
>   The following system variables are not updated during search for restart:
>   #5021 to #5028 ABSMT
>   #5041 to #5048 ABSOT
>   #5061 to #5068 ABSKP
> - DI/DO
>   During the restart of a program, DI can be read by a system variable, but DO cannot be output.
> - Clock
>   During the restart of a program, the clock time can be obtained by a system variable, but the time cannot be preset.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Quick program restart

5.7.1 Suppress Motion of Quick Program Restart

Overview

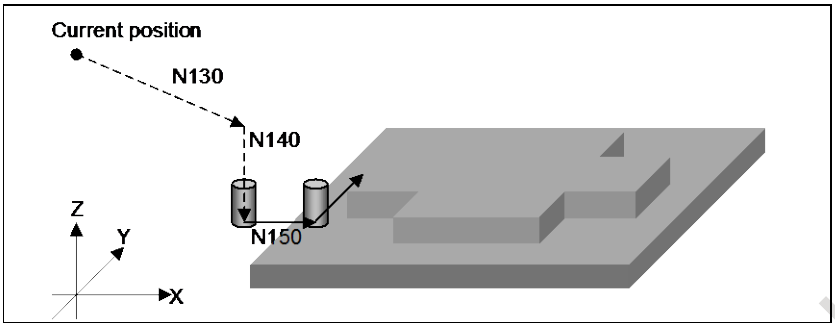
Suppress motion is a function which prohibits the approach to the start point of the specified restart block. In the conventional specification, the approach is performed to the start point of the restart block. To perform the approach to the end point of the restart block, the next block must be specified as the restart block. This function enables the tool to be moved to the end point of the specified restart block.
For example, when a block for approach is inserted in the machining program, the block for approach can be specified as the restart block and the tool can be moved to the end point of the block.

> **NOTE**
> An approach block is used for moving the tool closer to the workpiece from an arbitrary position (such as a tool change position). Normally, all axes used for the interrupted machining are specified. An approach block may consist of more than one block. In this case, select the first approach block as the restart block.

Example) Operation example

```
O1234;
:
N100 G91G28Z0.;
N110 G28X0.Y0.;
N120 T01M6;
N130 G90G00X-50.Y0;          Approach block for the X- and Y-axes
N140 Z-10.;                  Approach block for the Z-axis
N150 G41G01X-50.F1000.D1;
N160 Y50.;
:
```

When N130 is specified as the restart block, the tool moves from the current position to the end point of N130.



Explanation

Bit 5 (SPR) of parameter No. 11250 can be used to select whether to use suppress motion. To use suppress motion, press soft key [SET SUPRES] on the program restart information screen.

- Block for which suppress motion is available

To perform suppress motion, select a block which satisfies conditions (1) to (3) below:

(1) An absolute command is specified in the block.
If an incremental command is specified in the restart block, alarm PS1930, "ILLEGAL COMMAND AFTER RESTART", occurs.

(2) The G00 or G01 command is specified in the block.
If a command other than G00 or G01 is specified in the block, alarm PS1930 occurs.

(3) The block is not a modal block listed below.
If the restart block is a modal block listed below, suppress motion is not available even when the G00 or G01 command is specified in that block. If suppress motion is performed for a modal block listed below, alarm PS1931, "ILLEGAL MODE AFTER RESTART", occurs.

- Programmable mirror image
- 3-dimensional coordinate system conversion
- Coordinate system rotation
- Tilted working plane indexing
- Tilted working plane indexing by tool axis direction
- Tilted working plane indexing (incremental multiple command)
- Scaling
- Polar coordinate interpolation
- Normal direction control
- Canned cycle for drilling
- Canned cycle

- Saving the suppress motion status

By setting bit 6 (SAV) of parameter No. 11250 to 1, the suppress motion status can be saved. With this function, when suppress motion is always executed, once suppress motion setting is performed, the setting can be omitted after that.

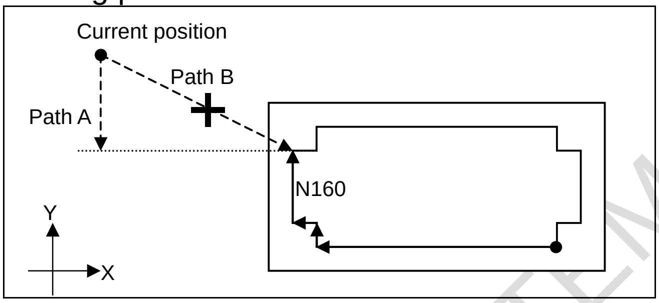
The suppress motion status can be checked using parameter No. 7310. When suppress motion is used, a negative value is set in parameter No. 7310.

For an axis for which an invalid value is set in parameter No. 7310, the suppress motion status is not saved.

Caution

⚠ **CAUTION**

When suppress motion is used, the tool moves to the end point of the restart block only along the axis specified in the restart block. For this reason, the tool does not move to the end point of the restart block along any axis that is not specified in the restart block.
When N160 in the example is specified for the restart block, the tool moves not along path B, but along path A as shown below:



Along any axis that is not specified in the restart block, move the tool to a position where it will not hit anything when it moves during restart operation, in the manual mode in advance.

Note

NOTE

1 While the tool is moving along another axis, suppress motion cannot be set or canceled. If suppress motion is set or canceled while the tool is moving along another axis, the warning message, “COMMAND ILLEGAL USE”, is displayed.
2 Performing approach for each arbitrary axis in the suppress motion status cancels the suppress motion.

Parameter

7310	Ordinal number of an axis which a movement is made in dry run after program restart

[Input type] Setting input
[Data type] Byte axis
[Valid data range] 1 to (Number of controlled axes)

This parameter sets the ordinal number of an axis along which a movement is made in dry run after the program is restarted.

⚠ **WARNING**

1 When there is two or more axes which has the same setting value in this parameter, the movement to the restart point is not executed in the second or more axis.
2 When the quick program restart is effective and bit 6 (SAV) of parameter No.11250 is set to 1, the movement to the restart point is not done in the axis which this parameter is set negative value(-1 to -(Number of controlled axes)).

NOTE

When this parameter is 0, alarm SR5020, "PARAMETER OF RESTART ERROR" or the warning "ORDINAL NUMBER ERROR (RESTART)" is displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
11250		SAV	SPR					

[Input type] Parameter input
[Data type] Bit path

#5 SPR Suppress motion is:
0: Disabled.
1: Enabled.

#6 SAV The suppress motion state is:
0: Not saved to a parameter.
1: Saved to a parameter.

Alarm and message

Number	Message	Description
PS1930	ILLEGAL COMMAND AFTER RESTART	The restart block does not satisfy either of the following conditions: (1) An absolute command is specified in the block. (2) The G00 or G01 command is specified in the block. Select a block satisfying conditions (1) and (2) as the restart block.
PS1931	ILLEGAL MODE AFTER RESTART	Suppress motion is specified in a mode in which suppress motion is not available. Select a block in a mode in which suppress motion is available as the restart block.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Suppress motion of quick program restart

5.7.2 Quick program restart for multi path system

Overview

When the machining was restarted by quick program restart in multi path system, it has been necessary to operate the program restart every one path.
In this function, when the program restart is operated in one path, the program restart is automatically operated as for other paths. As a result, the program restart can be executed by an easy operation in the machine with a multi path system.

Explanation

When the program restart is operated in certain one path, it is automatically operated in the multi path system as for other paths.
This operation is called "multi path quick program restart" ("multi path restart" at the following).
As for this function, only the program restart of the search method is possible. It cannot be used with the direct jump method.

Setting

In the path that sets the same group number as parameter No.7337, Multi path restart is available.
When the program restart is operated in a certain path, it is operated at the same time in the path that belongs to the same group.
In the path that sets 0 to parameter No.7337, Multi path restart is not available.

Blocks in which restart is possible

It is the following blocks that can restart the machining by the multi path restart.

(1) Block that is memorized in program restart memory, and was interrupted machining

(2) Block of waiting M code that was memorized in program restart memory, or arbitrary waiting M code specified on program restart setting screen

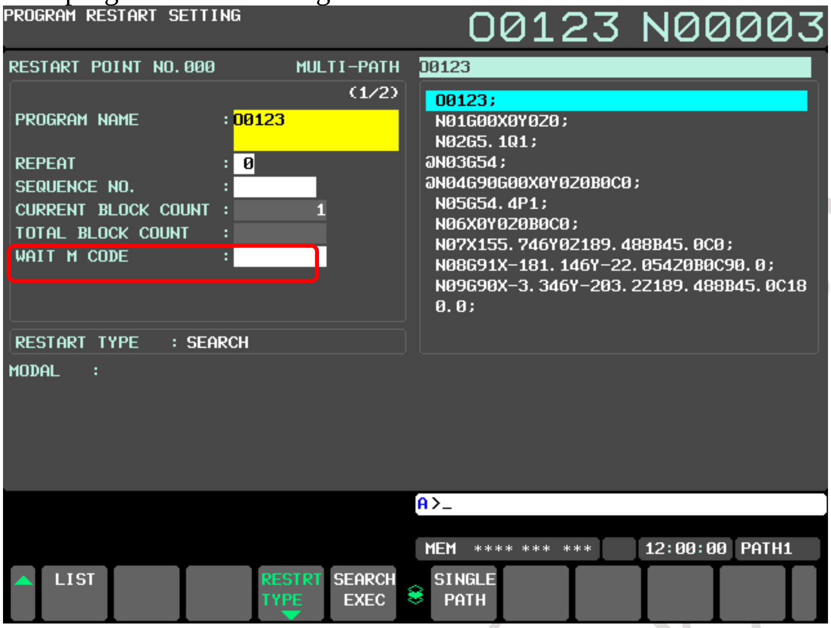


Fig. 5.7.2 (a) Screen for specify waiting M code

(3) Block of arbitrary sequence number specified on program restart setting screen without using program restart memory

When the program restart is executed to the block like the above-mentioned in certain one path, an operation of the program restart is automatically started as for other paths. When an operation of the program restart is started, each path starts the search of a target block severally.

Operation when the block of above-mentioned (1) - (3) is specified is as follows.

(a) Block where machining was interrupted
The block where machining is interrupted is searched for in each path and the search stops at the block.

(b) Block of the waiting M code
A block of the specified waiting M code is searched in each path, and the search stops in a block in which it was found. However, if the subprogram call is commanded in the same block as the waiting M code, the search stops at a first block of the subprogram. (Machining is restarted from a head of subprogram.)

(c) Block of arbitrary sequence number
A block of the specified sequence number is searched in each path, and the search stops in a block in which it was found. This method can be used in case that the same sequence number is put on the program in each path as a sign of the delimitation of the machining.

When the block that corresponds to above-mentioned (a)-(c) is specified as a restart block, the following method to specify are possible.

Fig. 5.7.2 (a) Search that restart for multi path system is possible

Method to specify / Kind of restart block	Select in list or detail screen	Select in list or detail screen and edit	Direct specification by newly edit
(a) Block where machining was interrupted	Available	Not available	Not available
(b) Block of the waiting M code	Available	Not available	Available
(c) Block of arbitrary sequence number	Not available	Not available	Available

NOTE

1. Warning "CAN NOT RESTART IN MULTI PATH MODE" is displayed when multi path restart is executed with specifying blocks other than the above-mentioned (a) to (c).
2. No-wait signal NOWT <G0063.1> and NMWT <Gn063.7> can be used as well as normal automatic operation when searching a waiting M code. It is not searched in a path which the waiting M code is disable.
   However, warning "CAN NOT RESTART IN MULTI PATH MODE" is displayed in the following cases.
   - It is ignoring waiting in operation path
   - The path that can be searched by ignoring waiting is below one path.
3. When the alarm is issued in a path executing the multi path restart, alarm (PS0060) "SEQUENCE NUMBER NOT FOUND" or alarm (PS0071) "DATA NOT FOUND" is issued in other paths of the same group.

Operation after search is completed

The target path of dry run movement after search is specified by bit 0 (MPD) of parameter No.7331.

- When parameter MPD is set 0 :

The order in the path which dry run movement is executed after search can be specified by parameter No.7338.

It enters the state of STRT after the axes move to the machining restart position with dry run and the machining program becomes waiting for execution.

When all paths of the same group enter the state of STRT, the waiting state is released and the machining program is executed.

In the path that 0 is set to parameter No.7338, the dry run movement and the machining program are executed without waiting for other path.

- When parameter MPD is set 1 :

Dry run is executed in the path that has been selected by the path selection signal HEAD<G0063.0>, HEAD2<G0062.7>.

It is necessary to do cycle start in each path whenever the selected path is changed by the path selection signal.

It enters the state of STRT after the axes move to the machining restart position with dry run and the machining program becomes waiting for execution.

When all paths of the same group enter the state of STRT, the waiting state is released and the machining program is executed.

NOTE

1. In case of bit 0 (MPD) of parameter No.7331 is set 0, when one path enters in the state of STRT, the movement of a dry run of next path is automatically started.
2. In case of bit 0 (MPD) of parameter No.7331 is set 1, the path has not been selected by the path selection signal does not accept the cycle start to a dry run movement after search.
3. Program restart under way signal SRNMV<Fn002.4> becomes "1" until the movement of a dry run in all paths that belong to the same group is completed. However, when the movement of a dry run in the path is completed, signal SRNMV<Fn002.4> becomes "0" as for the path set to parameter MPD=0 and parameter No.7338=0.

⚠ **WARNING**

The path of the same group to restart a multi path begins the execution of the machining program after waiting for other paths, but it does not synchronize with the path that is not the same group.

Single path restart

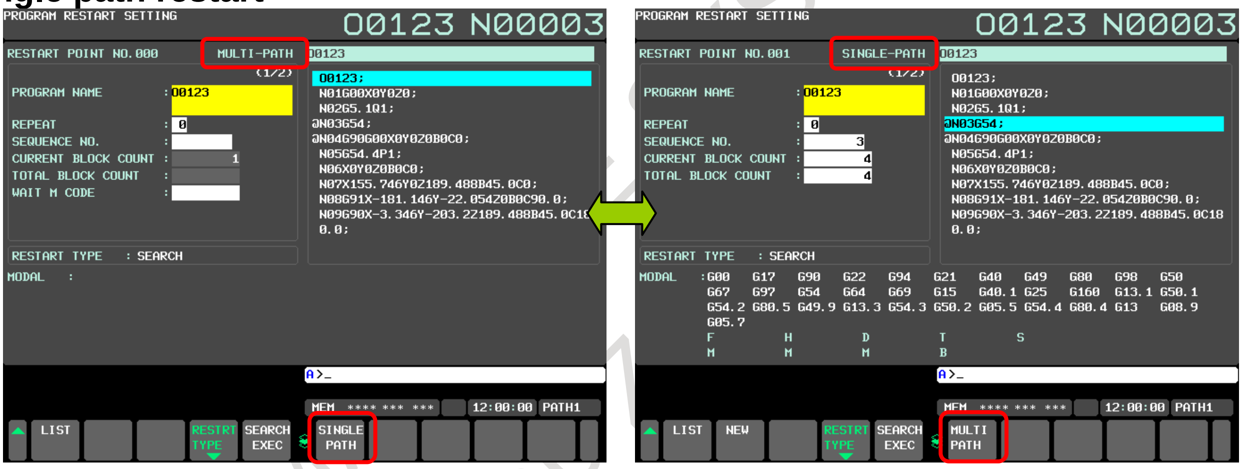


Fig. 5.7.2 (b) Program restart screen

When a soft key [SINGLE PATH] is pushed, it is change in the single path restart mode.

In the single path restart mode, it can be done the program restart operation only in the path under the display.

"SINGLE PATH" is displayed on the right of a left screen like showing in Fig. 5.7.2 (b) the single system restart mode inside.

The single path restart mode is canceled by either of the following operations.

- Program restart operation is executed.
- Another screen was selected.
- Soft key [MULTI PATH] is pushed.

Limitation

- Miscellaneous function

The miscellaneous function under the search is not output to PMC.

Please restore the state of the miscellaneous function by MDI operation before restarting the machining after the search ends.

- Synchronous / Composite / Superimposed control

On/off command of the synchronous control, the composite control, and the superimposed control by the miscellaneous function or G code under the search is invalid.

Please restore the state of the synchronous control, the composite control, and the superimposed control by MDI operation before restarting the machining after the search ends.

- Flexible path axis assignment

In the command of the flexible path axis assignment under the search, the flexible path axis assignment is not operated.
Please restore the state of the flexible path axis assignment by MDI operation before restarting the machining after the search ends.

Parameter

The parameter used by this function is shown as follows.

	#7	#6	#5	#4	#3	#2	#1	#0
7331								MPD

[Input type] Setting input
[Data type] Bit

#0 MPD In the multi path quick program restart function, the target path for a dry run movement after restart search is :
0: Specified by parameter No.7338.
1: Specified by path select signal HEAD<G0063.0> and HEAD2<G0062.7>.

> **NOTE**
> When MPD is set 1, do the cycle start further after the axes moves to the machining restart position to make each path stand-by in the STRT state. In the multi path quick program restart, all paths in the same group should be these stand-by states for an automatic operation to be executed.

7337	Group number of multi path quick program restart

[Input type] Setting input
[Data type] Byte path
[Valid data range] 0 to (Number of controlled paths)
Please set the group number to this parameter in the path which the multi path quick program restart is effective.

Example) In the four paths system, when path1 to 3 and path4 are made each group, please set this parameter as follows.

Path	Setting value of parameter No.7337
1	1
2	1
3	1
4	2 (or 0)

In this setting, the program restart is automatically executed in path2 and path3 set to the same group when the program restart is executed in path1.
Similarly, when the program restart is executed in path2 or path3, the program restart is automatically executed in two remaining paths.
When the program restart is executed in path4, the program restart is executed only in path4.

> **NOTE**
> If a value that is bigger than the number of controlled paths is set, warning "CAN NOT RESTART IN MULTI PATH MODE" is displayed at the search.

7338	Ordinal number of a path when moving by dry run in multi path quick program restart

[Input type] Setting input
[Data type] Byte path
[Valid data range] 0 to (Number of controlled paths)

Set the order of the path when moving to the restart point by a dry run after it begins to restart. When the same number is set, it moves at the same time. The path to which 0 is set does not wait for other paths, and the movement of a dry run and the machining program are executed.

Example) In the four paths system, to move a dry run in order in the first path2, in the second path1 and path3, at the end path4, set this parameter as follows.

Path	Setting value of parameter No.7338
1	2
2	1
3	2
4	3

> **NOTE**
> This parameter is effective at bit 0 (MPD) of parameter No.7331 is set 0.

5.7.3 Improvement of Quick program restart for non-machining program

Overview

In Quick Program Restart function, when a program is executed in MEM or RMT mode, restart block information is automatically memorized in the program restart memory.
After a machining program is interrupted, another program, which is not related to machining, is sometimes executed for the purpose of tool retract and the preparation of the machining in MEM or RMT mode. In this case, restart the block information is newly memorized, and the restart block information on the machining program memorized till then is overwritten. (It is not overwritten when a non-machining program is executed in MDI mode.)
By this improvement, restart block information is not memorized in the case of a certain specific program.
As a result, even when programs of tool retract and machining preparation are executed during the machining, restart block information related to machining can be kept.

Explanation

When the parameter to activate this function is set and the following programs are executed, restart block information is not memorized.

(1) Program with specific program number
Specify the range of the program number by the parameter No.7335 and No.7336. When the program with the program number specified here is selected as a main program, restart block information is not memorized.
Moreover, restart block information of a subprogram called from this main program is not

memorized. When the program specified here is called from the main program whose restart block information is memorized, restart block information is memorized.

NOTE
The folder cannot be used. Restart block information is not memorized when the program number is within the range, regardless of a folder.

(2) One touch macro call
Restart block information is not memorized by setting bit 0 (OMC) of parameter No.7330=1 while a one-touch macro is being executed. Restart block information of a subprogram called from one touch macro is not memorized, either.

Example

If parameter No.7335=7900 and No.7336=20 are set when program O7900-O7919 is used as a program for the tool retract, the relation between the main program number and the memorization of restart block information is as follows.

Main program number	Restart block information
O0001 — O7988	Memorized
O7900 — O7919	Not memorized
O7920 — O9999	Memorized

For instance, when O0001 is executed, restart block information about O0001 is memorized.
Restart block information about O0001 is kept, even if O0001 is interrupted on the way and O7901 for tool retract is selected as a main program and executed.
The program can be easily restarted by using kept restart block information if O0001 is selected again as a main program after the tool retract program is executed.
Restart block information about O0001 is overwritten, when O0001 is interrupted on the way and O0002 is newly executed as a main program, and restart block information on O0002 is memorized.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7330								OMC

[Input type] Parameter input
[Data type] Bit path

#0 OMC While executing One-touch macro, restart block information on Quick program restart is :
0: Memorized.
1: Not memorized.

7335	The first O number of program not memorized in program restart memory

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 99999999

Set the first number of the program that is not memorized as restart block information in quick program restart function.

7336	The number of programs not memorized in program restart memory

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 32767

Set the number of the programs that is not memorized as restart block information in quick program restart function.

Example)
When parameter No.7335=7900 and No.7336=20 are set, the program of O7900-O7919 (20 programs) is treated as a program not memorized in the program restart memory. When these programs are executed as the main program, restart block information is not memorized.

5.7.4 Improvement of Storing in Program Restart Memory

Overview

In Quick program restart, the storing the block information in the program restart memory can be disabled by the input signal of PMC or the system variable.

Explanation

In this function, the storing the block information in the program restart memory can be disabled by the input signal of PMC or the system variable. Therefore, the storing of useless block information as a restart point can be controlled, and useful restart points can be stored in the program restart memory more.

Signal

The storing the block information in the program restart memory can be disabled by program restart memory storing disabled signal QRSTD<Gn517.6>.
When this signal is set to “0”, block information is stored in the program restart memory.
When this signal is set to “1”, block information is not stored in the program restart memory.
If the state of QRSTD is “1” when an automatic operation begins, the program restart memory is not cleared and the storing in the program restart memory is disabled until end of the automatic operation.

System variable

The storing the block information in the program restart memory can be disabled by the system variable #151171 of custom macro.
When #151171 is set to 0, the block information is stored in the program restart memory.
When #151171 is set to 1, the block information is not stored in the program restart memory.
#151171 is cleared to 0 by a reset operation. When an automatic operation begins, the program restart memory is cleared without depending on #151171.

System variable number	System variable name	Attribute	Description
#151171	[#_QRSTD]	R/W	The control to store the block information in the program restart memory.

Use program restart memory storing disabled signal QRSTD<Gn517.6> in an automatic operation that should not use program restart function.
The system variable #151171 is effective when the storing the block information in the program restart memory temporarily is disabled while the program execution.

Example

<When you would like to execute the program restart after another program is executed once after the processing interrupts>

1. Set program restart memory storing disabled signal QRSTD<Gn517.6> to “1”.
2. Another program O1000 is selected, and executed.
3. Set program restart memory storing disabled signal QRSTD<Gn517.6> to “0”.
4. Program O0001 when the processing interrupts is selected because restart block information when the processing interrupts remains,
   and program restart is executed.

This usage is effective for the program name (Program name is not a number or DNC operation) that cannot be set to parameter No.7335 and No.7336.

<When you would like to disable to store in the program restart memory partially of a program>

In program O0002, when you would like to disable to store in the program restart memory during executing program O9001 called by a macro call, the program is made as follows.

```
O0002 ;                         O9001 ;
G28 X0.0 ;                      #151171=1 ;      It disables to store in the program restart memory.
Mxx ;                           #100=#1+#2 ;
:                               G00 G91 X#100 ;
G65 P9001 L2 A1.0 B2.0 ;              :          This section is not stored in the program restart memory.
:                               Myy ;
M30 ;                           G00 G91 Y#100 ;
                                #151171=0 ;      It enables to store in the program restart memory.
                                M99 ;
```

It is possible to substitute by program restart memory storing disabled signal QRSTD<Gn517.6>, but the program can be executed at high speed or more by using the system variable if storing and not storing restart memory are switched in the program.

Signal

Program restart memory storing disabled signal QRSTD <Gn517.6>

[Classification] Input signal
[Function] It is selected whether to store a block information in the program restart memory.
[Operation] When this signal is set to "0", block information is stored in the program restart memory.
When this signal is set to "1", block information is not stored in the program restart memory.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn517		QRSTD						

Alarm and message

Number	Message	Description
PS0110	OVERFLOW :INTEGER	The tolerance of the system variable was exceeded.

Note

- Use M code without buffering when you change program restart memory storing disabled signal QRSTD <Gn517.6 > during an automatic operation. Moreover, whether the block of this M code is stored in program restart memory is decided by the state of the signal when read in advance.
- System variable #151171 can be written only 0 or 1. When other values are written, the alarm (PS0110) "OVERFLOW :INTEGER" occurs.

5.7.5 Manual Intervention in Quick Program Restart

Overview

In Search method of Quick program restart, tool moves to the restart position automatically. This movement is executed in the specified order of axes one by one. Traditionally, if manual intervention is

executed in this movement to avoid interference with mechanical elements and others, reset and selection of restart block have to be executed again.
By manual intervention in Quick program restart, the movement to restart position can be continued without reset and selection of restart block even if operation mode change or manual intervention is executed before the movement is completed.

Explanation

Manual intervention in Quick program restart is enabled by setting the bit 5 (RMI) of parameter No.11251 to 1. If RMI is set to 0, Quick program restart works traditionally.

Traditional specifications (RMI is set to 0)

If RMI is set to 0, manual intervention is executed before the movement to restart position is completed, reset and selection of restart block have to be executed again.

Specifications when this function is enabled (RMI is set to 1)

If RMI is set to 1, operation mode is changed before the movement to restart position is completed, the movement to restart position is executed again from the first axis. Therefore, the movement can be continued without reset or selection of restart block, even if manual intervention is executed before the movement is completed.

Example

Example when this function is enabled is following.

(1) Assume that tool is retracted while N2 block is executing.
In this case, restart position is set to the starting position of N2 block when operation is restarted from interrupt position.

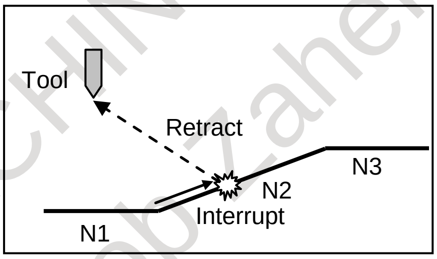


Fig. 5.7.5 (a) Starting of Quick program restart

(2) After reset is executed, N2 block is selected as the restart block and Quick program restart is executed. The movement to restart position is executed per axis in the specified order. Assume that operator notices possibility of interference when the movement of first axis is completed, and moves the tool to the other position by manual intervention.

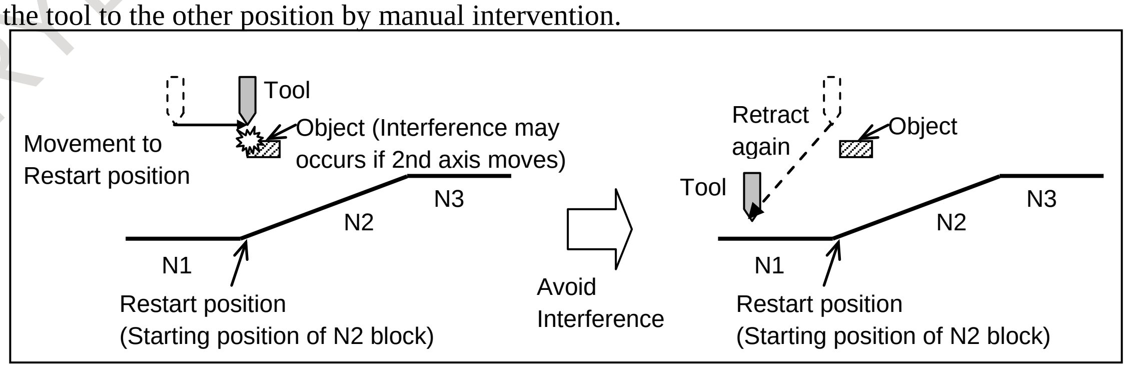


Fig. 5.7.5 (b) Interference avoidance by manual intervention

(3) After tool retraction of (2), the tool moves to the restart position correctly even if without reset because the movement to restart position is executed again from first axis.

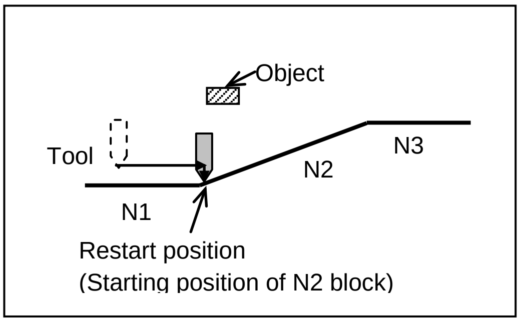


Fig. 5.7.5 (c) Movement to restart position after manual intervention

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11251			RMI					

[Input type] Parameter input
[Data type] Bit

#5 RMI Manual intervention in Quick program restart is :
0: Disabled.
1: Enabled.

Restriction

In case that bit 5 (RMI) of parameter No.11251 is set to 1, modal information is not restored again if it is changed in MDI mode during interruption of movement to restart position. If modal information have to be restored again, reset and selection of restart block is needed again.

5.7.6 Tool axis direction approach at program restart

Overview

In Program restart and Quick program restart (Search type), when the program restart is done to the block of the following function executing, the restart operation of the tool axis direction becomes possible.
This function is valid by bit 2 (TLR) of parameter No.7301 is set to 1. By the parameter (No. 7311), the approach distance to restart operation starting position of the tool axis direction can be set.

NOTE
1. This function requires "Program restart" function or “Quick program restart” function. To enable "Program restart" function, set NSQ (bit 2 of parameter No.8135) to 0.
2. This function is possible only for the tool rotation or mix type machine (the parameter No. 19680 is set to 2 or 21).

Explanation

- Moved to the restart position

A series of operation example of restart operation of the tool axis direction is shown in Fig. 5.7.6 (a) and Fig. 5.7.6(b). Moreover, Table 5.7.6 (a) shows the order of restart operation and the content.

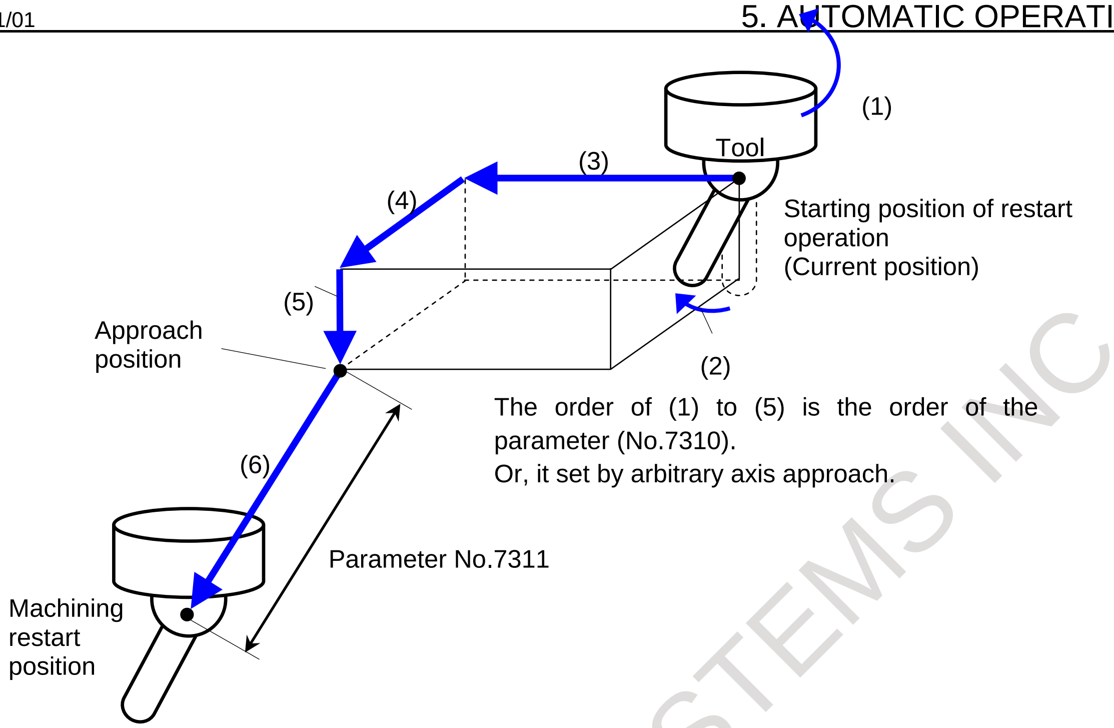


Fig. 5.7.6 (a) Restart movement of tool axis direction (parameter TLR=1) - example 1

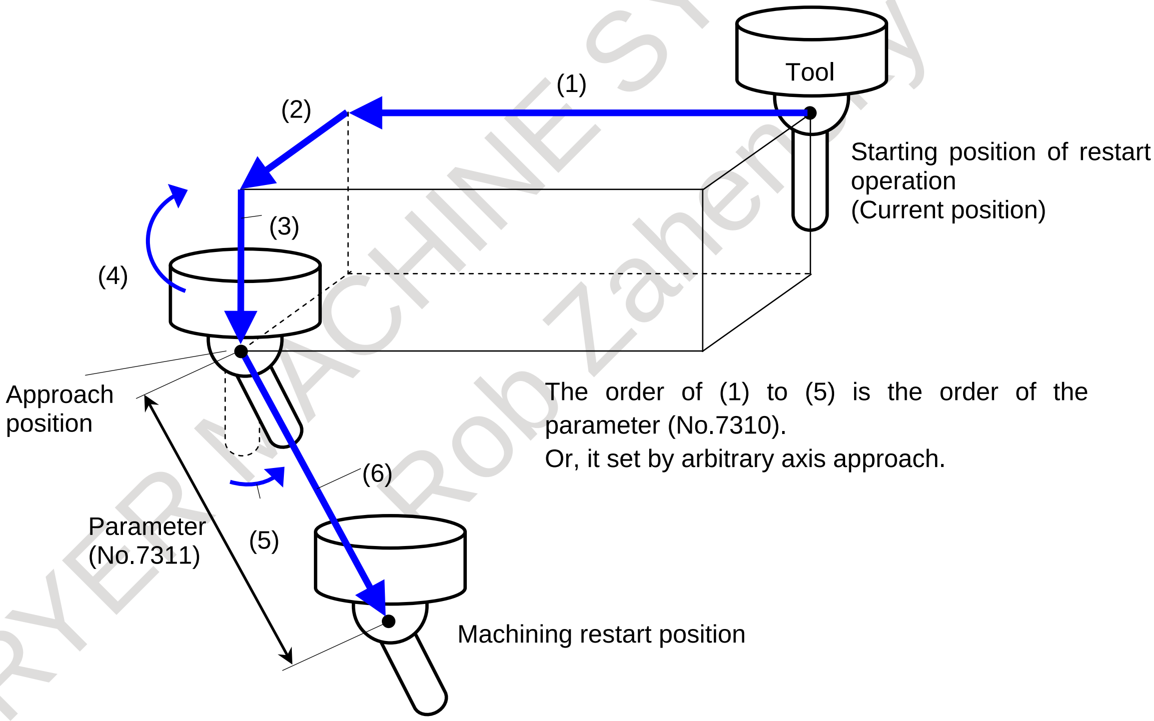


Fig. 5.7.6 (b) Restart operation of tool axis direction (parameter TLR=1) - example 2

Table 5.7.6 (a) Flow of restart movement of tool axis direction

Order	Contents of restart operation	Note
(1)-(5)	The following axes are positioned in order by the parameter (No. 7310) or arbitrary 1-axis approach	
	- Restart operation of rotary axis	- Positioning of the rotary axis becomes a base of the tool axis direction
	- Positioning to approach position with linear 3-axis	- The approach position is a point where the parameter (No. 7311) leaves the machining restart position for the tool axis direction
	- Restart operation of other axis	
(6)	Positioning to machining restart position as tool axis direction	

- Feedrate of restart operation

Restart operation is moved at the dry run rate.

- Tool axis direction approach

In case of that the tool position (position of rotary 2-axis and linear 3-axis) is already at the machining restart position when restart operation start, the tool axis direction approach is invalid.

- Single block

When the single block signal SBK <Gn046.1> set to 1, the single block stop is done in each of restart operation, approach position and machining restart position of each axis.

- Functions that cannot be used simultaneously

- Suppress motion of quick program restart
- Quick program restart for multi path system

Parameter

7310	Ordinal number of an axis along which a movement is made in dry run after program restart

[Input type] Setting input
[Data type] Byte axis
[Valid data range] 1 to (Number of controlled axes)

This parameter sets the ordinal number of an axis along which a movement is made in dry run after the program is restarted.

> ⚠ **WARNING**
> 1 When there is two or more axes which has the same setting value in this parameter, the movement to the restart point is not executed in the second or more axis.
> 2 When the quick program restart is effective and bit 6 (SAV) of parameter No.11250 is set to 1, the movement to the restart point is not done in the axis which this parameter is set negative value(-1 to -(Number of controlled axes)).

> **NOTE**
> When this parameter is 0, alarm SR5020, "PARAMETER OF RESTART ERROR" or the warning "ORDINAL NUMBER ERROR (RESTART)" is displayed.

19680	Mechanical unit type

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 21

Specify the type of the mechanical unit.

Parameter No. 19680	Mechanical unit type	Controlled rotation axis	Master and slave
2	Tool rotation type	Two rotation axes of the tool	The first rotation axis is the master, and the second rotation axis is the slave.
21	Mixed type	One rotation axis of the tool + one rotation axis of the table	The first rotation axis is the tool rotation axis, and the second rotation axis is the table rotation axis.

5.8 TOOL RETRACT AND RECOVER

Overview

The tool can be retracted from a workpiece to replace the tool, if damaged during machining, or to check the status of machining. Then, the tool can be returned to restart machining efficiently.

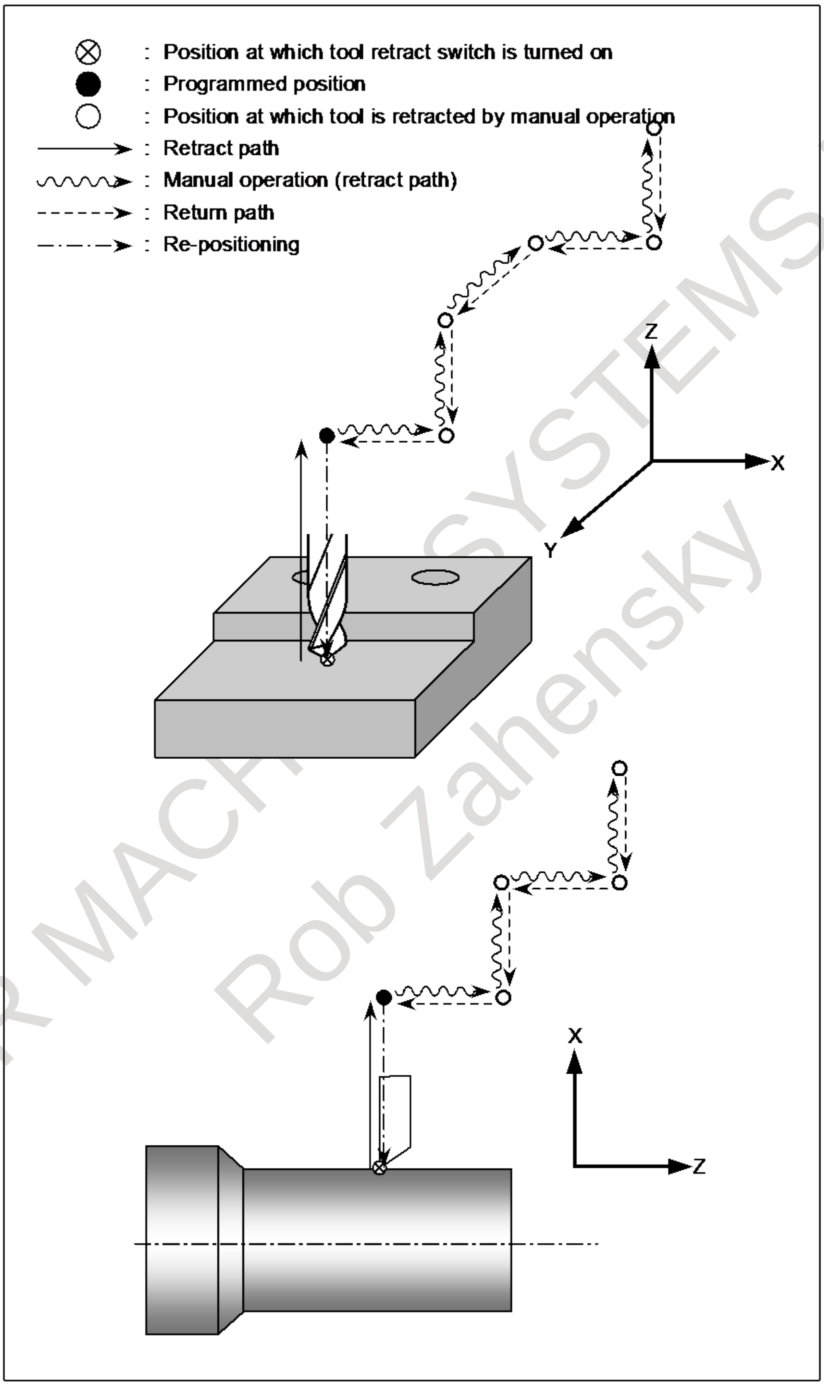


Fig. 5.8 (a) Path of tool retract and recover for milling machining

- Basic procedure for tool retract and recover

(1) When tool retraction signal TRESC <Gn059.0>, which is an input signal from the PMC, is set to 1 during automatic operation, in the automatic operation stopped state, or in the automatic operation halted state, the tool is retracted by a specified distance. This operation is called retraction. The position at which the retraction is completed is called the retract position.

The tool is retracted by linear interpolation at the minimum feedrate among the parameter (No.7042) values of moving axes.
If a block is being executed halfway when the TRESC signal is set to “1” during automatic operation, the execution of the block is interrupted immediately, then retraction is performed. After the retraction is completed, the automatic operation is halted. If G10.6 is not programmed, retraction does not take place, block execution is interrupted, and the automatic operation is halted or stopped.
The distance of retraction can also be specified in parameter No. 7041.
When the TRESC signal is set to “1”, it is said that the tool retraction mode is entered.
The tool retraction mode involves all of retraction, manual retraction, recovery, and re-positioning operations.
When the tool retraction mode is entered, tool retraction mode signal TRACT <Fn092.3> is set to “1”, notifying the PMC that the tool retraction mode is set.
When the tool starts moving to retract, tool retraction axis movement signal TRMTN <Fn092.4> is set to “1”. When the tool is stopped at the retract position, TRMTN is set to “0”, and tool retraction/return completion signal TRSPS <Fn092.5> is set to “1”.

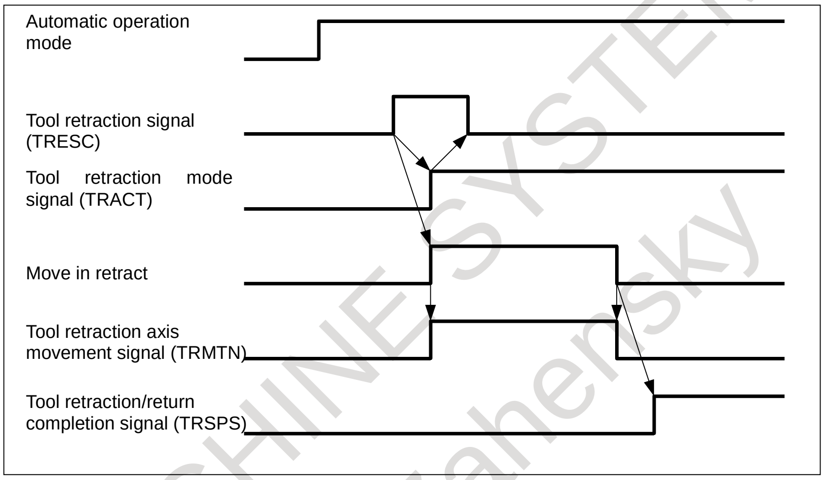


Fig. 5.8 (b) Timing chart of retraction operation (when movement is involved)

(2) Change the mode to the manual mode, and move the tool by a manual operation (jog feed, incremental feed, manual handle feed) for a purpose such as the purpose of changing the tool. This operation is called manual retraction. The CNC automatically records the retract path of the tool. Up to 20 points on the tool retract path are recorded automatically. When the tool is retracted manually, the TRSPS signal is set to “0”. While the tool is moving along an axis by manual operation, the TRMTN signal is “0”.

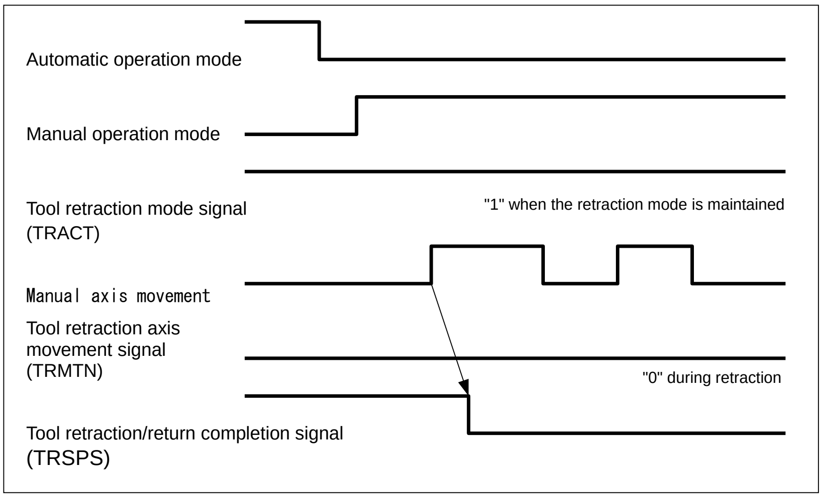


Fig. 5.8 (c) Timing chart of manual retraction

NOTE
If the path number is 5 or over, the maximum number of record points on the tool retract path is 10.

(3) Restore the automatic operation mode, and set tool return signal TRRTN <Gn059.1> to “1” then back “0”. The CNC then reverses the tool along the path of the manual movement to return the tool to the retract position. This operation is called recovery. When the tool starts moving for recovery, TRMTN is set to “1”. Upon completion of the recovery to the retract position, TRSPS is set to “1”. The recovery is performed at the feedrate specified by parameter No.7042 for each axis.

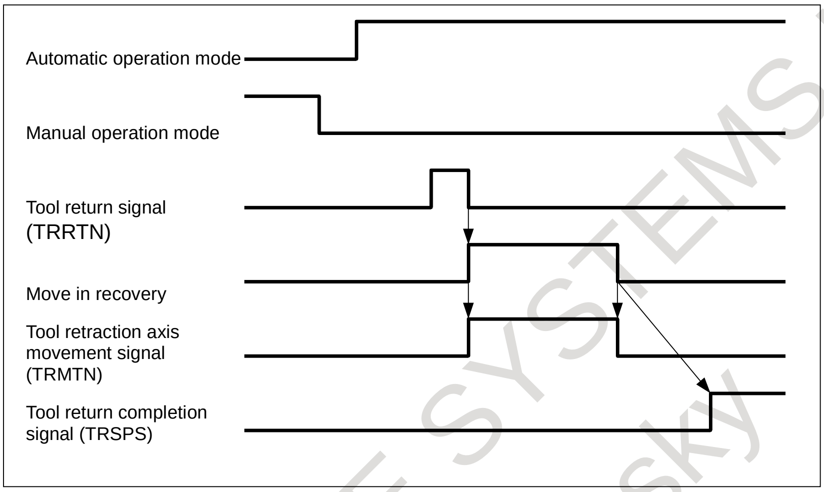


Fig. 5.8 (d) Timing chart of recovery (when the tool returns to the retract position without performing single block stop or feed hold stop during recovery)

(4) When a cycle start is directed at the retract position, the tool first moves to a position at which tool retraction signal TRESC was set to “1”. This operation is called re-positioning. As re-positioning starts, tool retraction/return completion signal TRSPS is set to “0”. During movement for re-positioning, tool retraction axis movement signal TRMTN is “1”.
The re-positioning movement is made by linear interpolation at the minimum feedrate among the parameter (No.7042) values of moving axes.
Upon completion of the re-positioning, tool retraction mode signal TRACT is set to “0”, notifying the PMC that the tool retraction mode ends. The subsequent operation varies depending on the automatic operation state present when the tool retraction mode was entered as follows:
<1> When the tool retraction mode was entered during automatic operation, the interrupted automatic operation is restarted continuously after re-positioning is completed.
<2> When the tool retraction mode was entered from the automatic operation stopped state or automatic operation halted state, the automatic operation stopped state or automatic operation halted state is once entered, respectively, after re-positioning is completed. Then, directing a cycle start restarts automatic operation.

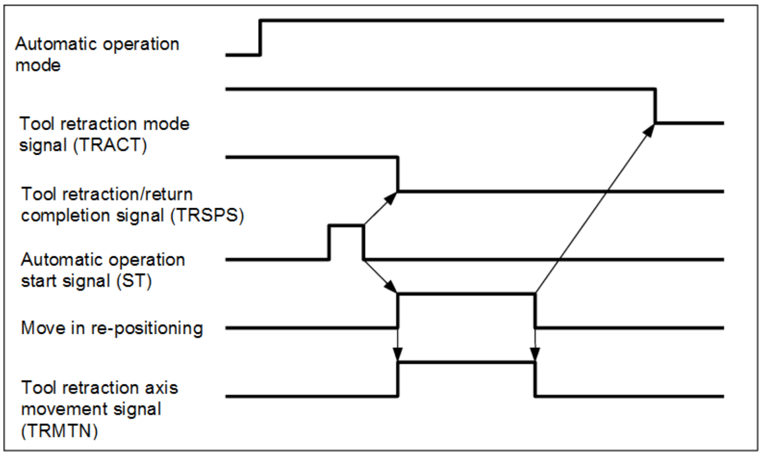


Fig. 5.8 (e) Timing chart of re-positioning (when movement is involved)

NOTE

1 If negative value or zero is set in parameter (No.7042) of moving axes, retract, recovery and re-positioning are performed at the dry run feedrate.

2 Manual feedrate override signal is effective for the feedrate of parameter (No.7042).

- Tool retract and recover for threading

Tool retract and recover for threading is different from ordinary tool retract and recover as follows

(1) During retraction, chamfering is performed between the specified retraction axis and threading axis. Furthermore, in this state, tool retraction axis movement signal TRMTN is not set to “1”

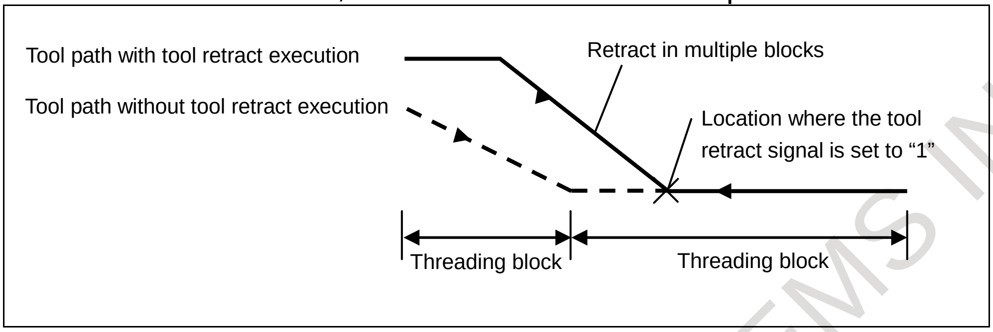
(2) After retraction, one block that does not specify threading is executed and the tool stops.

(3) When the major axis for threading is specified as the retraction axis, retraction is not performed by turning the TOOL WITHDRAW switch on. In this case, after a block that does not specify threading is executed, an alarm PS0429, “ILLEGAL COMMAND IN G10.6” is issued and the tool stops.

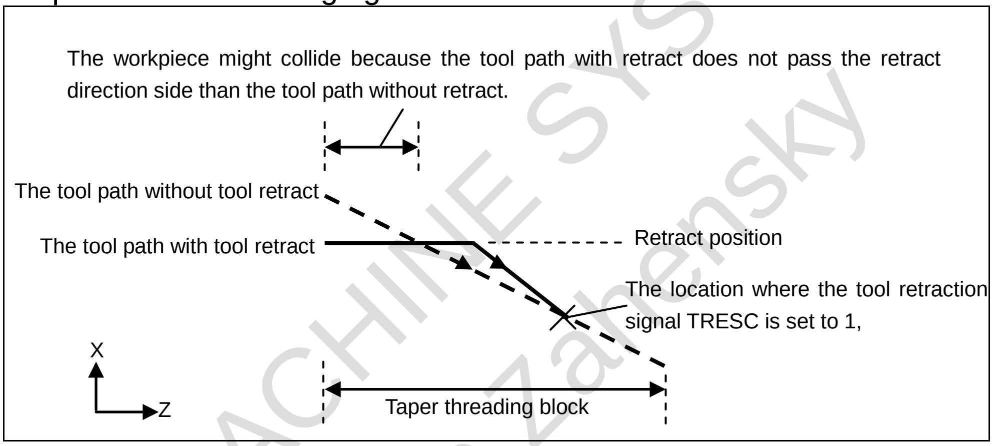
(4) As repositioning, the tool is returned to the position specified in the first block that does not specify threading.

NOTE

3 In threading cycle and multiple threading cycle, tool retract and recover is invalid.
4 In circular threading, tool retract and recover is invalid.
5 When dry run is executed, tool retract and recover for threading is invalid.
6 In continuous threading blocks other than last threading block, if retract start at a location near the block end, retract is executed in multiple blocks.



7 In taper threading, specify the retract position as the tool does not collide the workpiece surely because the tool might collide depending on specification of retract position as following figure.



Signal

Tool retraction signal TRESC<Gn059.0>

[Classification] Input signal

[Function] Tool retraction mode is selected.

[Operation] When this signal is turned to "1", the CNC retracts the tool by a pre-programmed distance.

Tool retraction mode signal TRACT<Fn092.3>

[Classification] Output signal

[Function] This signal reports that tool retraction mode is set. When the CNC is reset while the signal is "1", the signal is turned to "0", and tool retraction mode is canceled.

[Output cond.] The signal is turned to "1" when:
- Tool retraction mode is selected.

The signal is turned to "0" when:
- Tool retraction mode is not selected.

Tool return signal TRRTN<Gn059.1>

[Classification] Input signal

[Function] In tool retraction mode, a tool that has been retracted manually along an axis is returned to the retract position along the same axis.

[Operation] When this signal is turned from “1” to “0”, the CNC traces back the path of the manually moved tool to automatically return the tool to the retract position.

Tool retraction axis movement signal TRMTN<Fn092.4>

[Classification] Output signal

[Function] This signal reports that the tool is moving along an axis during retraction, recovery, or re-positioning.

[Output cond.] The signal is turned to “1” when:

- The tool moves along an axis during retraction, recovery, or re-positioning.

The signal is turned to “0” when:

- The tool does not move along any axis during retraction, recovery, or re-positioning.

Tool return completion signal TRSPS<Fn092.5>

[Classification] Output signal

[Function] This signal reports that the tool is in the retract position in tool retraction mode. When this signal is “0”, re-positioning cannot be performed by pressing the cycle start button.

[Output cond.] The signal is set to “1” when:

- Retraction has been completed.
- The tool has been returned to the retract position.

The signal is set to “0” when:

- The tool is not in the retract position in tool retraction mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn059							TRRTN	TRESC

	#7	#6	#5	#4	#3	#2	#1	#0
Fn092			TRSPS	TRMTN	TRACT			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7040					TRC	RPS	TRS	TRI

[Input type] Parameter input

[Data type] Bit path

#0 TRI The G10.6 command for tool retract and return is:

0: Assumed to be an absolute or incremental programming according to the absolute or incremental programming mode.

1: Always assumed to be an incremental programming.

#1 TRS After the completion of repositioning in tool retract and return:

0: Automatic operation is restarted.

1: Operation stops when the single block switch is on. When a cycle start is executed again, automatic operation is started.

#2 RPS When the tool retract signal TRESC <Gn059.0> is set to “1” after G10.6 is specified alone:

0: The tool is not retracted.

1: The tool is retracted with the value set for parameter No. 7041 or 11261 used as the incremental retraction distance.

#3 TRC When automatic operation is restarted after the tool retract and return is executed during the execution of a drilling canned cycle:

0: Machining of the same cycle is performed again (the same drilling is performed).

1: Machining of the next drilling cycle is performed (the next drilling is performed).

7041	Retraction distance in tool retract and return

[Input type] Setting input
[Data type] Real axis
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the retraction distance used when G10.6 is specified alone for the tool retract and return. The tool is retracted by the distance set for this parameter in the incremental mode. This data is valid only when bit 2 (RPS) of parameter No. 7040 is set to 1.

7042	Feedrate for each axis in tool retract and recover

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)

Set the feedrate for each axis in tool retract and recover. The recovery is performed at the feedrate specified by this parameter. The retract and repositioning are performed at the minimum feedrate among the parameter values of moving axes. Furthermore, if maximum number of record point in manual retract is exceeded, the tool move to last record point by linear interpolation at the minimum feedrate among the parameter values of moving axes. Manual feedrate override signal is effective for the feedrate of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
10410								NRT

[Input type] Parameter input
[Data type] Bit axis

#0 NRT In tool retract and recover or manual intervention and return, the axis is:

0: Subject to tool retract and recover or manual intervention and return.

1: Not subject to tool retract and recover or manual intervention and return.

Limitation

Tool retract and recover cannot be performed along axes that are in the modes below.

- Superimposed control

Warning

> ⚠ **WARNING**
> The retraction axes and retraction distances specified with G10.6 need to be changed in appropriate blocks depending on the figure to be machined. An incorrectly specified retraction distance may damage a workpiece, the machine, or the tool. So, be very careful when specifying a retraction distance.

Alarm and message

Number	Message	Description
DS0310	NOT ON RETURN POINT	The return position recorded during retraction is not reached during recovery. The position may be displaced during recovery due to a machine lock or mirror image. Perform the operation again after making a reset.
PS0015	TOO MANY SIMULTANEOUS AXES	A move command was specified for more axes than can be controlled by simultaneous axis control. Either add on the simultaneous axis control extension, or divide the number of programmed move axes into two blocks.
PS0429	ILLEGAL COMMAND IN G10.6	When retraction was started in the threading block, the retract command was performed in the longitudinal direction of the threading.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Tool retract and recover

5.8.1 Improvement of Tool Compensation for Tool Retract and Recover

Outline

In this function, when the recovery operation or re-positioning operation is started, the updated compensation value is used. Therefore the restart operation is performed with the updated compensation value.

Explanation

If this function is effective (the bit 7 (TRO) of parameter No.7002 is set to 1), when the recovery operation or re-positioning operation is started, the updated compensation value is used and the restart operation is performed with the updated compensation value. Thus, if the compensation value is updated after exchanging the tool at the manual retract position or the retract position, the tip point of tool becomes the same position as it of before exchanging tool in the restart operation thereafter.

The operation that updated compensation value is effective can be selected by the bit 6 (TNR) of parameter No.7002.

Example-1 (When the bit 6 (TNR) of parameter No.7002 is set to 0):
Assuming that the compensation value is updated from OFS_1 to OFS_2 after exchanging the tool at the manual retract position. When the recovery operation is started, the updated compensation value OFS_2 is used, the tip point of new tool and old tool is the same position in the retract position. (Refer to the figure Fig. 5.8.1 (a))

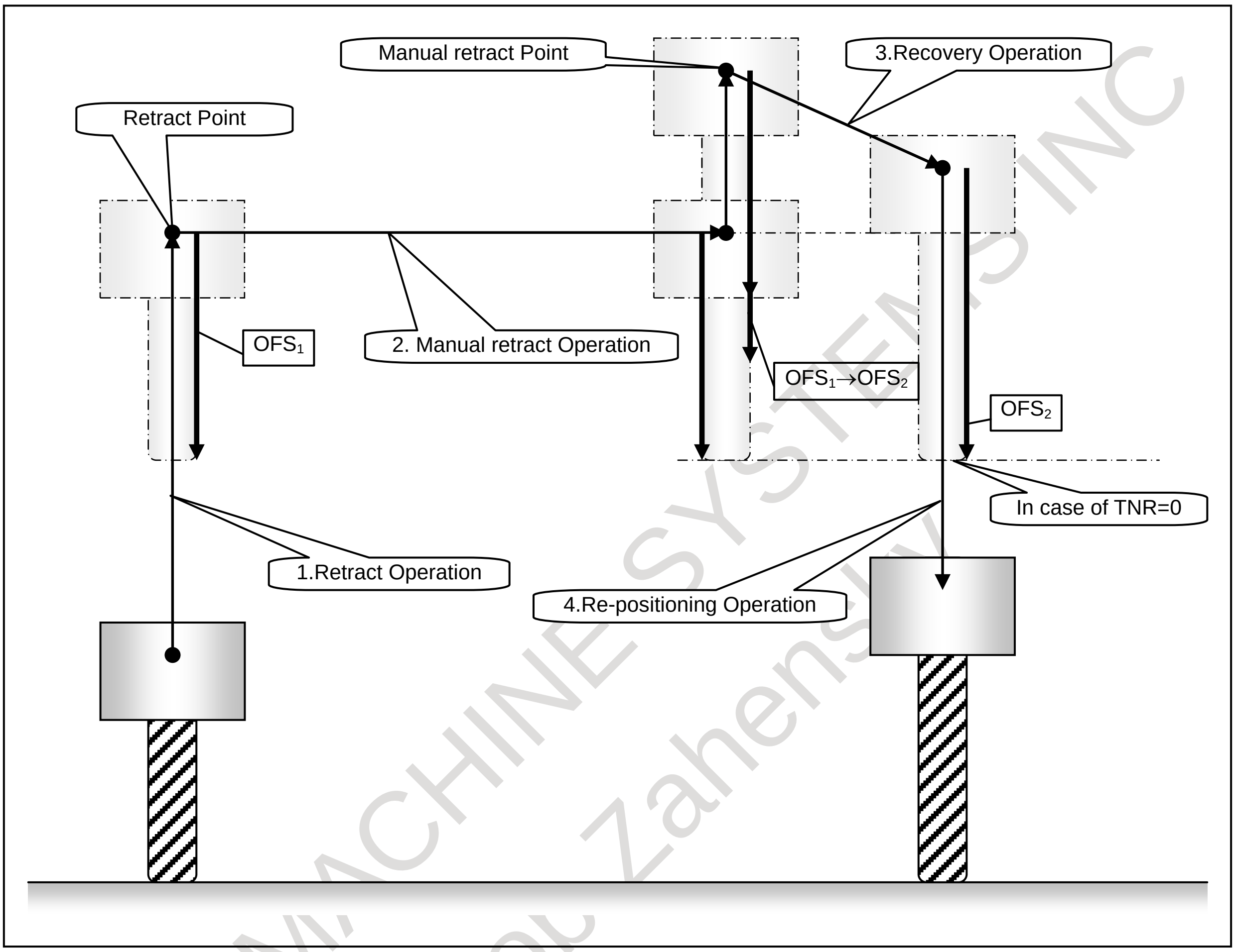


Fig. 5.8.1 (a)

Example-2 (When the bit 6 (TNR) of parameter No.7002 is set to 1):
Assuming that the compensation value is updated from OFS_1 to OFS_2 after exchanging the tool in the manual retract position. When the recovery operation is started, the updated compensation value OFS_2 is not used. After re-positioning operation, the updated compensation value OFS_2 is used (Refer to the figure Fig. 5.8.1 (b))

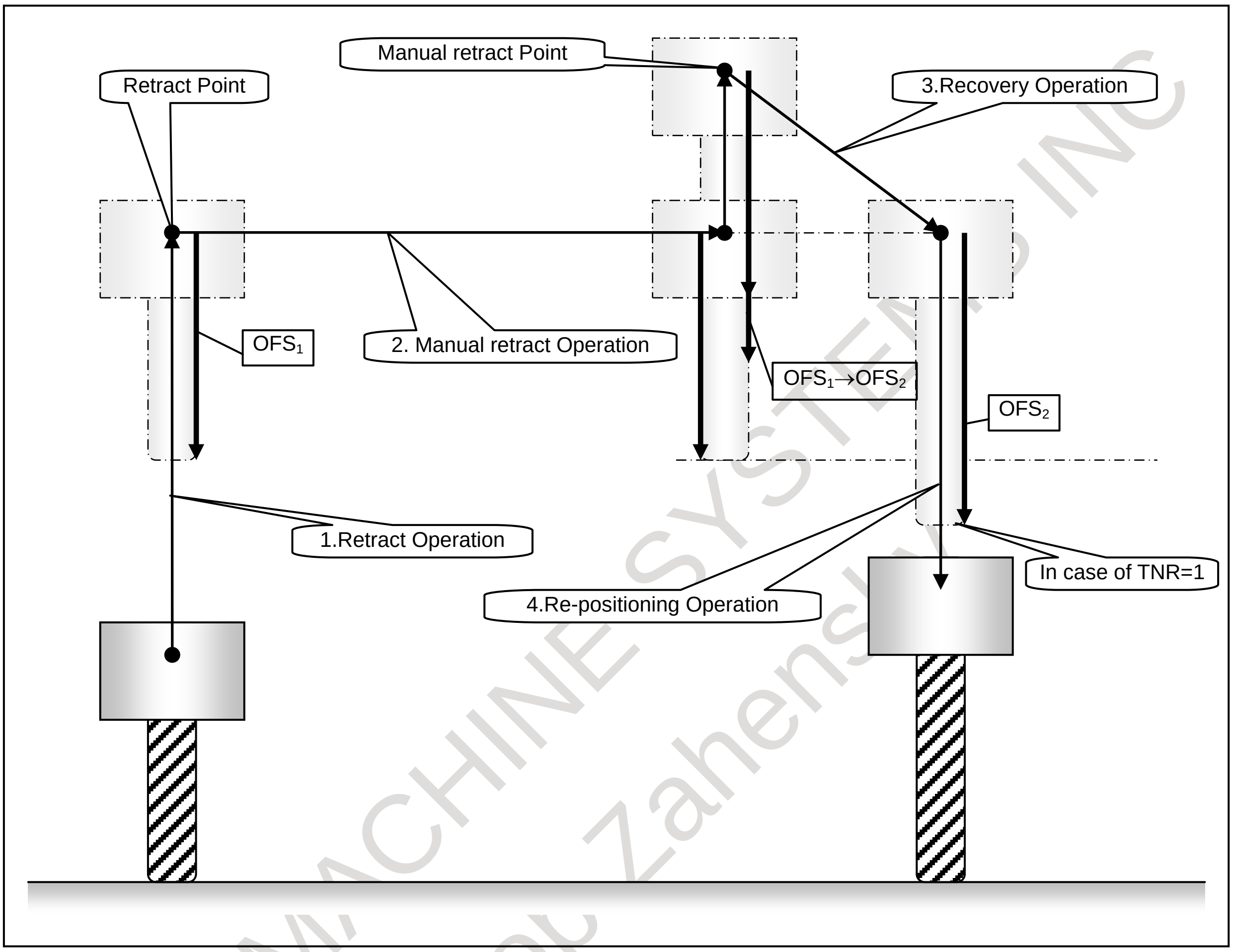


Fig. 5.8.1 (b)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7002	**TRO**	**TNR**						

[Input type] Parameter input
[Data type] Bit path

#6 TNR When the updated compensation value in the tool retract and recover function is effective (the bit 7 (TRO) of parameter No.7002 is set to 1)
0: The updated compensation value is effective in the recovery operation.
1: The updated compensation value is effective in the re-positioning operation.

#7 TRO When the compensation value is updated while the tool retract and recover function is executing,
0: The updated compensation value is invalid.
1: The updated compensation value is effective.

Notes

NOTE

1 The compensation considered by this function is as follows. Neither the cutter compensation nor the tool nose radius compensation is considered.
Machining system : Tool length compensation G43/G44 (wear/geometry),
Tool Offset G43.7/G44.7 (wear /geometry)
Lathe system : When the Extended Tool Selection is not used (the bit 3 (TCT) of parameter No.5040 is set to 0)
Tool compensation T code (wear /geometry)
When the Extended Tool Selection is not used (the bit 3 (TCT) of parameter No.5040 is set to 1)
Tool length compensation G43/G44 (wear /geometry),
Tool Offset G43.7/G44.7 (wear /geometry)

2 The compensation value can be changed only in the retract position or the manual retract position.

3 When the tool length compensation B is used, the vertical axis against the plane from which the tool retract and recover is executed is compensated. This plane is not a same plane when G43/G44 is instructed (In case machining system)

4 When the tool length compensation C is used and G43 or G44 is commanded, the compensated axis is indefinite. Therefore this function cannot be used in this case.

5 This function cannot be used during the movement to the reference position by the G28/G30 command or the G53 command is executing.

5.9 MANUAL INTERVENTION AND RETURN

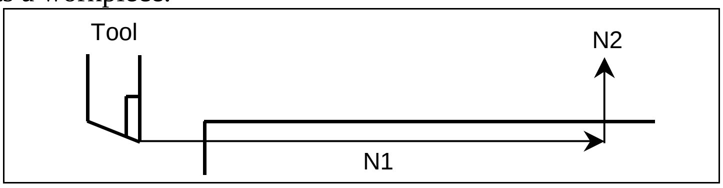
Overview

If you use feed hold to stop the tool from moving an axis during automatic operation and restarts the tool after manual intervention, for example, for checking a cutting surface, the tool can resume automatic operation after automatically returning to the pre-intervention position.
This function is optional.

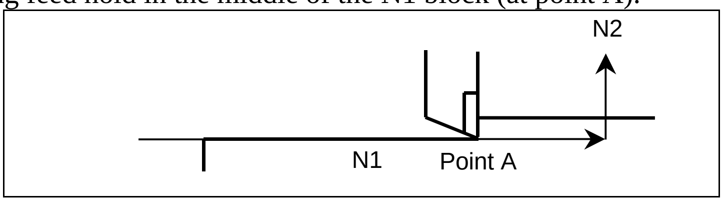
Explanation

Setting bit 0 (MIT) of parameter No. 7001 enables manual intervention and return.
The sequence of manual intervention and return is as follows:

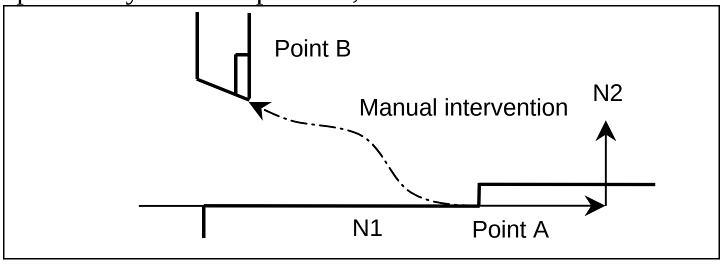
1. The N1 block cuts a workpiece.



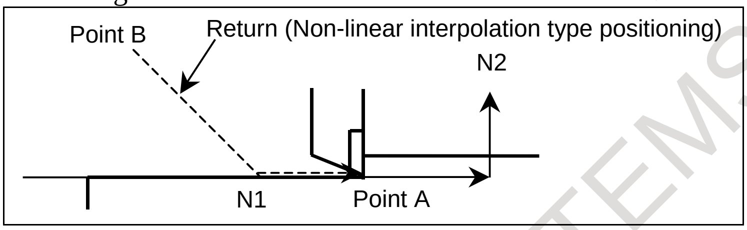
2. Stop the tool using feed hold in the middle of the N1 block (at point A).



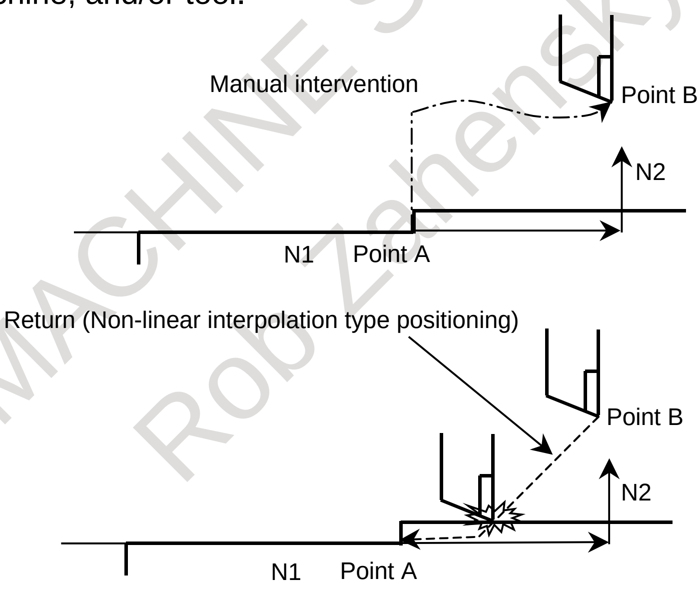
3. Retract the tool to point B by manual operation, then restart the machine.



4. After the tool automatically returns to point A by non-linear interpolation type positioning at the dry run feedrate, the remaining move command of the N1 block is executed.



> ⚠ **WARNING**
> Manual intervention must be performed correctly with meticulous care, following the machining direction and the shape of the workpiece, not to damage the workpiece, machine, and/or tool.



- Manual absolute on/off

In manual absolute off mode, the tool does not return to the stop position. Instead, the tool moves according to the manual absolute on/off function.

- Return feedrate

For the return operation, the dry run feedrate is used, and the jog feedrate override function is enabled. When manual rapid traverse selection signal RT <G019.7> is 1, the return feedrate is not dry run feedrate but rapid traverse rate.

- Return operation

Return operation is performed according to non-linear interpolation type positioning.

- Single block
If the single block stop switch is on during return operation, the tool stops at the stop position once and restarts movement when cycle start is executed again.

- Cancellation
If a reset, alarm, or emergency stop occurs during manual intervention or return operation, the manual intervention and return function is cancelled.

- MDI mode
The manual intervention and return function can be used in the MDI mode as well.

- Relation to other functions
In other functions such as canned cycle, the manual intervention and return function is effective.

- PMC axis control
If intervention is performed under PMC axis control after feed-holding of automatic operation, return operation does not apply to the axis moving under PMC axis control when the return operation starts.
If the return operation is started with the axis under PMC axis control being stopped after having completed the PMC axis control command, however, the return operation is performed by the amount of movement by PMC axis control.
When PMC axis control is used for an axis unrelated to the program command, set bit 0 of parameter No. 10410 to 1 to exclude the axis completely from return operation by the manual intervention and return function.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7001								MIT

[Input type] Parameter input
[Data type] Bit path

#0 MIT Manual intervention and return function is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10410								NRT

[Input type] Parameter input
[Data type] Bit axis

#0 NRT In tool retract and recover or manual intervention and return, the axis is:
0: Subject to tool retract and recover or manual intervention and return.
1: Not subject to tool retract and recover or manual intervention and return.

Alarm and message

Number	Message	Description
PS5219	CAN NOT RETURN	Manual intervention and return cannot be performed during execution of 3-dimensional coordinate system conversion, tilted working plane indexing.

Limitations

- Enabling/disabling of manual intervention and return function

Manual intervention and return function is enabled in the feed hold state (feed hold lamp signal SPL<Fn000.4> is set to "1").
Manual intervention and return function is disabled in automatic operation stop state (single block stop, feed hold stop with no travel distance remaining, etc.) and the operation is performed according to the manual absolute on/off function.

- Offset

When the tool is replaced using manual intervention for a reason such as tool damage, even if the amount of offset is changed when restarting in the middle of the interrupted block, the change is not reflected in machining.

- Machine lock, mirror image, and scaling

When performing manual intervention and return, never use the machine lock, mirror image, or scaling function.

- 3-dimensional coordinate system conversion, tilted working plane indexing

Alarm PS5129 is issued when manual intervention and the return is attempted during execution of 3-dimensional coordinate system conversion, tilted working plane indexing.

- Parking

Do not switch the parking signal during manual intervention and return (for example, after feed hold stop). The tool may not return in the right position when the parking signal is switched during manual intervention and return.

- Manual intervention and return by multi-path system

If cycle start is executed at the same time on two or more paths in a multi-path system, the path that performed manual intervention does return operation. However, the path that did not perform manual intervention restarts processing without doing return operation.

- Axis for which return operation is not performed

When the axis is in the following conditions, return operation of the axis is not performed.

- Spindle positioning
- Tapping
- Slave axis of axis synchronous control
- Slave axis of synchronous control
- Spindle control by servo motor
- Polygon axis

> **NOTE**
> Under synchronous control, manual intervention and return can be performed for the slave axis only when bit 2 (PKUx) of parameter No. 8162 is 1 and the master axis is parking.

- Feed hold during return operation

Feed hold is not effective during return operation.

5.10 RETRACE

M

Overview

The tool can retrace the path along which the tool has moved so far (reverse execution). Furthermore, the tool can move along the retraced path in the forward direction (forward reexecution). After forward reexecution is performed until the tool reaches the position at which reverse execution started, machining is continued as programmed.

> **NOTE**
> This function is optional.

Procedure

- Forward execution → reverse execution

To perform forward execution of a program, set the reverse execution signal RVS<Gn007.0> to "0", then perform a cycle start operation. If the reverse execution signal RVS is set to "1", reverse execution or the end of reverse execution results.

To perform reverse execution of a program, use one of the following methods:

1) Set the reverse execution signal RVS to "1" during forward execution of a block.
2) Perform a single block stop operation during forward execution, then set the reverse execution signal RVS to "1".
3) Perform a feed hold stop operation during forward execution, then set the reverse execution signal RVS to "1".

When method 1) is used, reverse execution starts after the end of the block being executed (after execution up to the single block stop position). Reverse execution does not start as soon as the reverse execution signal RVS is set to on.

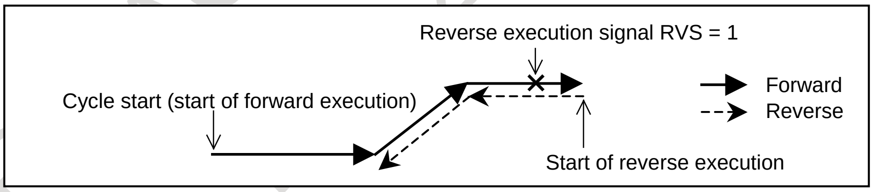


Fig. 5.10 (a)

When method 2) is used, performing a cycle start operation starts reverse execution from the position at which a single block stop takes place.

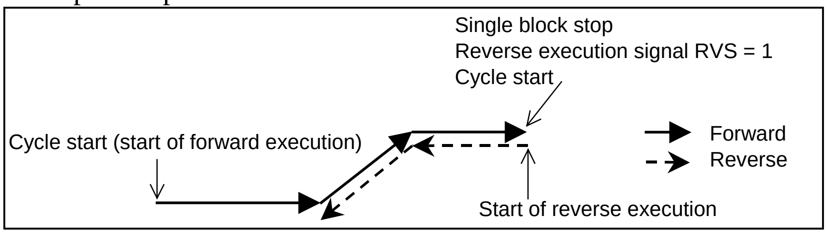


Fig. 5.10 (b)

When method 3) is used, performing a cycle start operation starts reverse execution from the position at which a feed hold stop takes place.

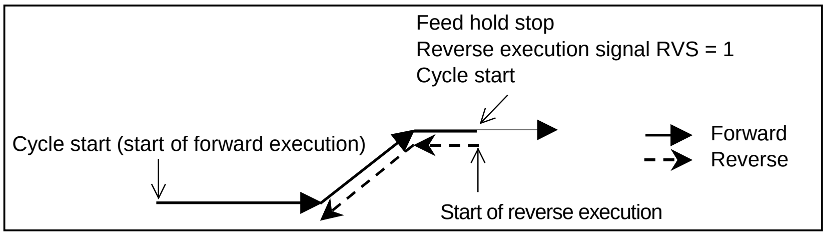


Fig. 5.10 (c)

- Reverse execution → forward reexecution

To perform forward reexecution of a program, use one of the following three methods:

1) Set the reverse execution signal RVS to 0 during reverse execution of a block.
2) Set the reverse execution signal RVS to 0 after a single block stop takes place during reverse execution.
3) Set the reverse execution signal RVS to 0 after a feed hold stop takes place during reverse execution.

When method 1) is used, forward reexecution starts after the block being executed ends (after execution up to the position at which a single block stop takes place). Forward reexecution does not start as soon as the reverse execution signal RVS is set to 0.

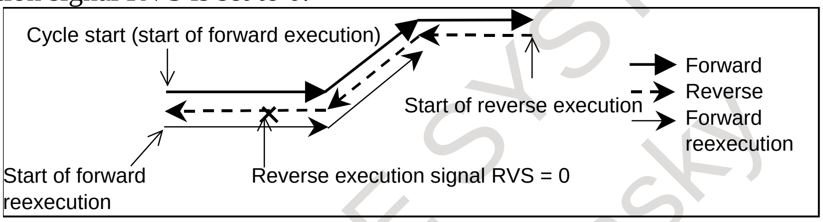


Fig. 5.10 (d)

When method 2) is used, performing a cycle start operation starts forward reexecution from the position at which a single block stop takes place.

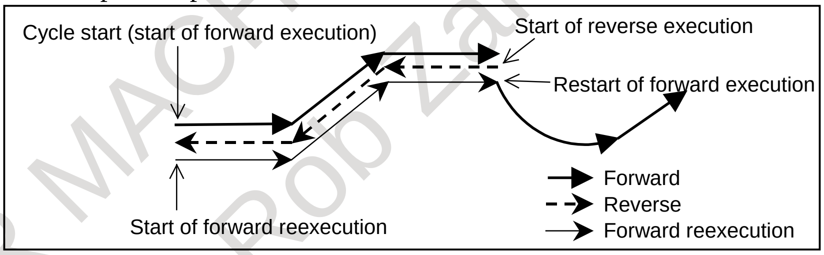


Fig. 5.10 (e)

When method 3) is used, performing a cycle start operation starts forward reexecution from the position at which a feed hold stop takes place.

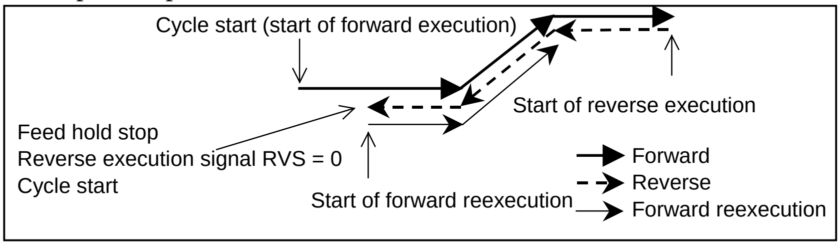


Fig. 5.10 (f)

- Reverse execution → end of reverse execution → forward reexecution

When a block to be executed is no longer present during reverse execution (when reverse execution has been performed up to the block where forward execution started, or when forward execution has not yet been performed), the reverse execution end state is entered, and operation stops.

Even when a cycle start operation is performed while the reverse execution signal RVS is held 1, operation is not performed, and the reverse execution end state is maintained. Forward reexecution (or forward execution) is started by setting the reverse execution signal RVS to 0 then performing a cycle start operation.

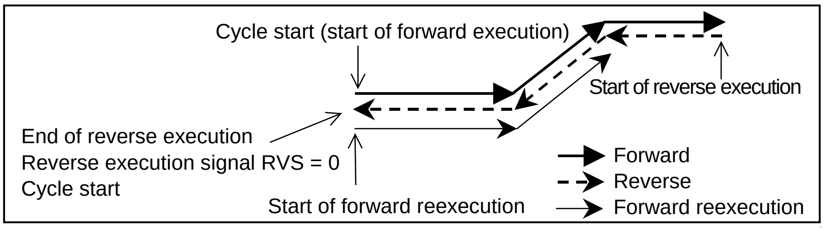


Fig. 5.10 (g)

- Forward reexecution → forward execution

After forward reexecution is performed up to the block at which reverse execution started, forward execution starts automatically, and commands are read from the program again and executed. No particular operation is required.

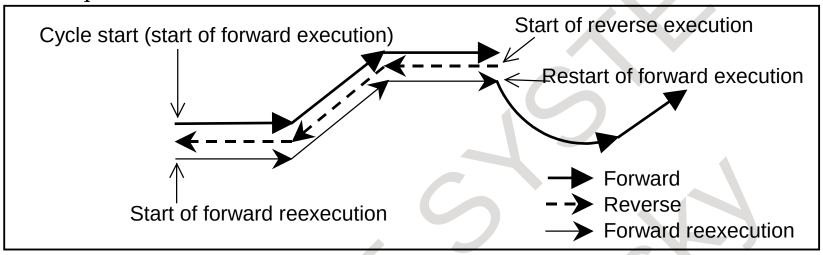


Fig. 5.10 (h)

If reverse execution was performed after feed hold stop, forward reexecution ends when the feed hold stop position is reached, then forward execution is performed. Also if single block operation was performed, forward reexecution ends at the single block stop position.

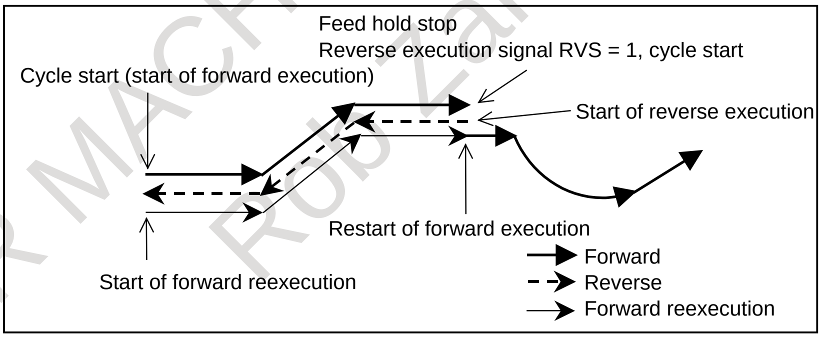


Fig. 5.10 (i)

Description

- Reverse execution and forward execution

Usually in automatic operation, a program is executed in the programmed order. This is called forward execution. This function allows a program executed by forward execution to be executed in the reverse direction. This is called reverse execution. Reverse execution allows the tool to retrace the path along which the tool has moved during forward execution.

Reverse execution of a program can be performed only for blocks that have been executed by forward execution.

Furthermore, in single block operation, reverse execution can also be performed on a block-by-block basis.

- Forward reexecution

Blocks that have been executed by reverse execution can be reexecuted in the forward direction up to the block from which reverse execution started. This is called forward reexecution. Forward reexecution allows the tool to retrace the same tool path as in forward execution until the position at which reverse execution started is reached.
After the block from which reverse execution started is reached, the program is executed again in the programmed order (forward execution).
Furthermore, in single block operation, forward reexecution can also be performed on a block-by-block basis.

- End of reverse execution

When a block to be executed is no longer present during reverse execution (when stored blocks have all been executed during reverse execution, or when forward execution has not yet been performed), operation stops. This is called the end of reverse execution.

- Status indication

During reverse execution, characters "RVRS" blink on the screen. During forward reexecution, characters "RTRY" blink to indicate that forward reexecution is in progress. The "RTRY" indication is kept blinking until the block at which reverse execution started is reached and normal operation starts (until forward execution is restarted).
When a block to be executed is no longer present during reverse execution, or if an attempt is made to perform reverse execution for a block that cannot be executed by reverse execution, characters "RVED" blink, notifying the user that reverse execution can no longer be performed.

- Number of blocks that can be executed by reverse execution

Up to about 100 blocks can be executed by reverse execution. Depending on the specified program, the maximum number of executable blocks may decrease.

- Reset

A reset operation (the RESET key on the MDI unit, the external reset signal, or the reset & rewind signal) clears the blocks stored for reverse execution.

- Feedrate

A feedrate to be applied during reverse execution can be specified in parameter No. 1414. If this parameter is set to 0, the feedrate in reverse execution is assumed to be the same as that in forward execution. Rapid traverse, however, is performed always at the rapid traverse rate, regardless of the setting of this parameter.
The feedrate in forward reexecution is always the same as that in forward execution.
In reverse execution or forward reexecution, feedrate override, rapid traverse override, and dry run are allowed.

- Start of reverse execution or forward reexecution after the end of a block

In a block for rapid traverse (G00), linear interpolation (G01), circular interpolation (G02, G03), dwelling (G04), skip cutting (G31), or an auxiliary function in an automatic operation mode (memory operation, part program operation, or MDI operation), reverse execution or forward reexecution can be started. However, reverse execution and forward reexecution do not start as soon as the reverse execution signal status is changed. When the block has ended, that is, after a movement, dwelling, or an auxiliary function is completed, reverse execution or forward reexecution starts.

- Start of reverse execution or forward reexecution after feed hold stop

When a feed hold stop operation is performed during execution of rapid traverse (G00), linear interpolation (G01), circular interpolation (G02, G03), or skip cutting (G31), then the reverse execution signal status is changed and operation is restarted, reverse execution or forward reexecution can be started immediately from the stop position. This cannot be performed when dwelling (G04) or an auxiliary function is being executed.

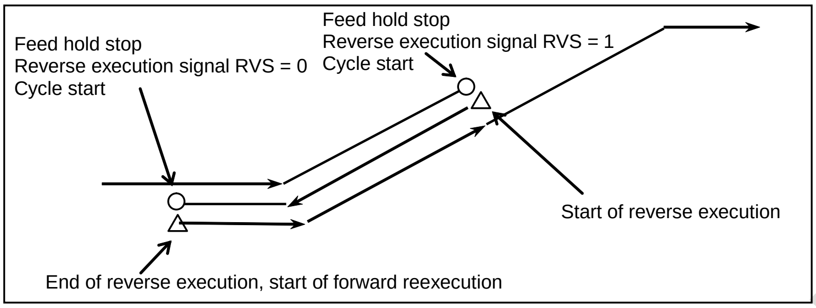


Fig. 5.10 (j)

When reverse execution is started after feed hold stop operation, the part from the start point of that block to the feed hold stop position is stored as one block. Therefore, when forward reexecution is performed with the single block switch set to 1, a single block stop takes place as soon as the position at which reverse execution started is reached.

- Start of reverse execution or forward reexecution after single block stop

After a single block stop takes place, reverse execution or forward reexecution can be started immediately when the reverse execution signal status is changed and restart operation is performed.

Limitation

- Blocks that cannot be executed by reverse execution

In the modes listed below, reverse execution cannot be performed.
When one of these commands appears during reverse execution, reverse execution ends immediately and "RVED" is displayed.

- Cylindrical interpolation (G07.1,G107)
- Polar coordinate command (G16)
- Functions related thread cutting (G33,G34,G35,G36)
- Polygon turning (G50.2, G51.2)
- Tool axis direction control (G53.1)
- Single direction positioning (G60)
- Tapping mode (G63)
- Tapping cycle (G84,G74)
- Rigid tapping cycle (G84,G74,G84.2,G84.3)
- Fine boring cycle (G76)
- Back boring cycle (G87)

It is impossible to perform reverse execution for blocks specifying the commands listed below. If one of these commands appears during reverse execution, reverse execution ends immediately and "RVED" is displayed.
Some of these commands turn a mode on and off. It is possible to start reverse execution and perform forward reexecution in a mode set by such a command. However, if a block that turns the mode on or off is reached during reverse execution, the reverse execution ends at that block, and "RVED" is displayed.

- Functions related AI contour control (G05,G05.1,G08)
- HRV3 on/off (G05.4)
- Inch/metric conversion (G20, G21)
- Stored stroke check on/off (G22, G23)
- Functions related reference position return (G27, G28, G29, G30)
- 3-dimensional coordinate system conversion (G68, G69)
- Feature coordinate system (G68.2)
- Figure copying (G72.1,G72.2)
- Index table indexing
- Cs contouring control

- Spindle positioning

- Switching of automatic operation mode

When a single block stop operation is performed during reverse execution or forward reexecution and the mode is changed from memory operation to MDI operation or vice versa, it becomes impossible to perform reverse execution, forward reexecution, and forward execution. To restart operation, restore the original mode, then perform a cycle start operation.

- Single block stop position

A block that is internally generated by the control unit is also treated as one block during reverse execution.

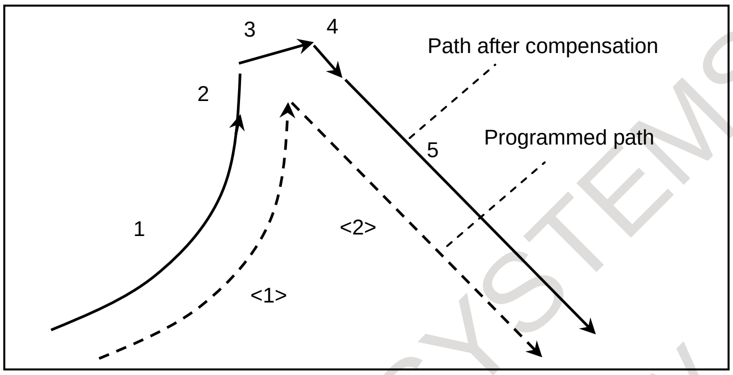


Fig. 5.10 (k) Path when cutter compensation is applied

In the above example, the program specifies two blocks, but in actual operation, move commands for five blocks are generated.
In such a case, positions at which a single block stop takes place may differ between forward execution and reverse execution.

- Positioning (G00)

When non-linear type positioning is performed (bit 1 (LRP) of parameter No. 1401 is set to 0), the tool path in reverse execution and that in forward execution do not match. The tool path in forward reexecution is the same as that in forward execution.
When linear type positioning is performed (bit 1 (LRP) of parameter No. 1401 is set to 1), the tool path in reverse execution is the same as that in forward execution.

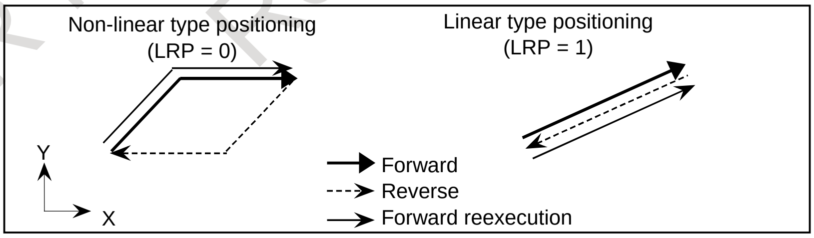


Fig. 5.10 (l)

- Dwell command (G04)

During reverse execution or forward reexecution, the dwell command (G04) is executed in the same way as in normal operation.

- Programmable data input (G10)

Tool compensation values, parameters, pitch error data, workpiece origin offsets, and tool life management values set or modified by programmable data input (G10) are ignored during reverse execution and forward reexecution.

- Skip function (G31) and automatic tool length measurement (G37)

The skip signal and the automatic tool length measurement signal are ignored during reverse execution and forward reexecution. During reverse execution and forward reexecution, the tool moves along the path that the tool has actually passed during forward execution.

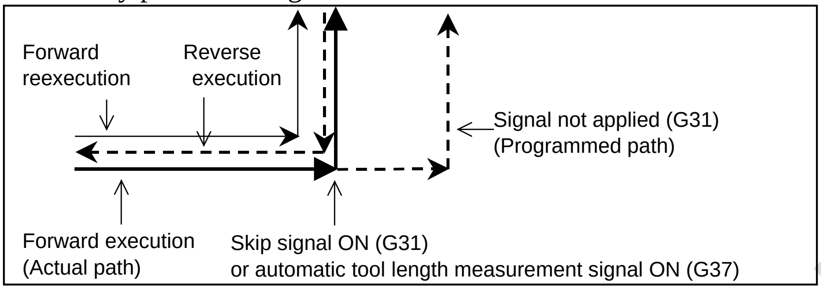


Fig. 5.10 (m)

- Setup of a coordinate system (G92, G54 to G59, G54.1P_, G52, and G92.1)

As for the display of the absolute coordinate value under reverse motion, the value in the work coordinate system when reverse motion begins is displayed. Therefore, when the command (G92, G54 to G59, G54.1P_, and G52) to which the work coordinate system is changed is passed, a different value might be displayed as forward movement. However, the actual machine position does not differ.

- Mirror image

When a block to which a mirror image is applied by programmable mirror image (G50.1, G51.1) is executed during reverse execution, the tool moves along the actual path resulting from the application of a mirror image in the reverse direction.
When a mirror image is applied to a block by setting or a machine signal, the block with the mirror image not applied is stored. Mirror image application by setting or a machine signal is enabled also during reverse execution and forward reexecution. Therefore, during reverse execution and forward reexecution, the mirror image by setting data or machine signal must be turned on and off so that this on/off status and the on/off status during forward execution match.

- Changing offsets

Even when cutter compensation offsets or tool length offsets are changed during reverse execution or forward reexecution, the change in compensation or offset data does not become valid until forward reexecution ends and normal operation starts. Until then, the tool moves with the offset data that was applied when the block was executed for the first time during forward execution.

- Feedrate clamp

During reverse execution or forward reexecution, feedrate clamp is not performed with parameter No. 1420 (rapid traverse rate) or parameters Nos. 1430 and 1432 (maximum cutting feedrate). It is executed with parameter No. 1414 or at the feedrate assumed during forward execution.
If, for example, the parameters above are set to smaller values during reverse execution or forward reexecution, clamp is not performed with these values, but with parameter No. 1414 or at the feedrate assumed during forward execution.
For clamp at the feedrate assumed during backward execution or forward reexecution, change the feedrate with the external deceleration or override signal.

- Interrupt type custom macro

(1) Do not initiate any interrupt during reverse execution.
(2) Do not execute an interrupted block and the interrupt program in reverse execution.

- Tool management function

The tool life is not counted during reverse execution and forward reexecution.

- Inverse time feed (G93)

If a nonzero value is set as the feedrate to be applied during reverse execution in parameter No. 1414, a block that moves the tool by inverse time feed during forward execution is executed at the parameter-set feedrate (feed per minute) during reverse execution.
If the feedrate during reverse execution (parameter No. 1414) is not set (= 0), the same feedrate as applied during forward execution is used.

- Maximum spindle speed clamp (G92Sxxxx)

Clamping at a maximum spindle speed specified during reverse execution becomes valid. This means that if G92Sxxxx appears during reverse execution, the spindle speed is clamped at Sxxxx in the subsequent reverse execution. As a result, the clamp speed may differ between reverse execution and forward execution even when the same block is executed. The spindle speed is clamped when the G96 mode is set.

- Auxiliary functions

M, S, T, and the second auxiliary function (B function) are output directly also during reverse execution and forward reexecution.
When specified together with a move command in the same block, M, S, T, and the second auxiliary function (B function) are output with the move command at the same time during forward execution, reverse execution, and forward reexecution. Therefore, the output positions of M, S, T, and the second auxiliary function (B function) during reverse execution differ from those during forward execution and forward reexecution.

- Custom macro

Custom macro operations are ignored during reverse execution and forward reexecution.

- Execution macro (macro executor)

Macro executor operations are ignored during reverse execution and forward reexecution.

- Tool retract and recover function

For retract operation and repositioning operation by the tool retract and recover function, reverse execution cannot be performed. Retract operation and repositioning operation are ignored during reverse execution and forward reexecution.

- AI contour control

When the reverse execution is started in AI contour control mode, a reverse execution ends immediately depending on a program then the reverse motion is not possible.
During reverse execution and forward reexecution, the feedrate clamp function by acceleration under AI contour control is disabled.

- Display

During reverse execution and forward reexecution, the modal display and the display of the currently executed program are not updated; information obtained at the start of reverse execution is maintained.

Signal

Reverse execution signal RVS<Gn007.0>

[Classification] Input signal

[Function] Requests the control unit during automatic operation (DNC operation and MDI operation except memory operation and binary operation) to return the tool by retracing the path the tool has passed so far.

[Operation] When this signal is set to “1” during forward execution, the machining path along which the tool has moved so far is retraced. Note that, however, reverse execution does not start as soon as this signal is set to “1”; reverse execution starts after the forward execution of the current block ends. When this signal is set to “0” during reverse execution, switching from reverse execution to forward reexecution takes place. Also in this case, forward reexecution does not start as soon as the signal is set to “0”, but forward reexecution starts after the reverse execution of the current block is completed.

To make swift switching from forward execution to reverse execution or from reverse execution to forward reexecution, set the automatic operation stop signal *SP to 0 to stop automatic operation, and after the automatic operation in-progress signal STL becomes “0” and the automatic operation stop state is entered, change the RVS signal status. Subsequently, after the automatic operation stop signal *SP is set to “1” and the status of the automatic operation in-progress signal ST is changed from “1” to “0” to start automatic operation, it becomes possible to change the execution mode to reverse execution or forward reexecution in the middle of a block.

Reverse execution in-progress signal RVSL<Fn082.2>

[Classification] Output signal

[Function] Posts that reverse execution is being performed.

[Output cond.] This signal is set to “1” when:

- The reverse execution signal RVS is set to “1” and reverse execution is being performed.

This signal is set to “0” when:

- The reverse execution signal RVS is set to “0” and forward reexecution or forward execution is being performed.
- Blocks to be executed are no longer present during reverse execution, and so execution is stopped.

During reverse execution, the M, S, T, and second auxiliary functions are executed in the same way as during forward execution. When these functions are not to be performed in the same way as during forward execution, use this signal to allow the PMC to perform appropriate processing.

In particular, when the M, S, T, or second auxiliary function and a move command are specified in the same block, the positions at which the code signal and strobe signal are output differ between forward execution and reverse execution. Therefore, use this signal and distribution completion signal DEN to allow the PMC to perform appropriate processing as necessary.

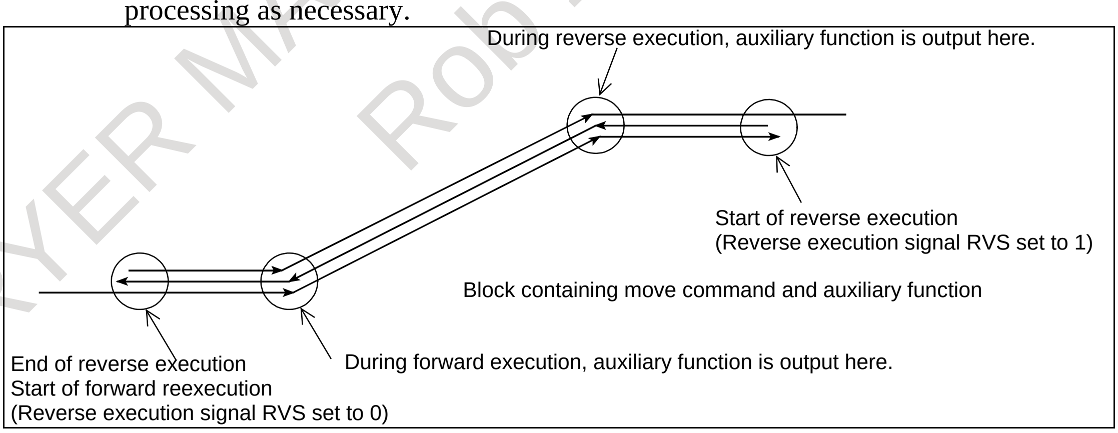


Fig. 5.10 (n)

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn007								RVS

	#7	#6	#5	#4	#3	#2	#1	#0
Fn082						RVSL		

Parameter

1414	Cutting feedrate for reverse motion

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set a cutting feedrate for retrace operation. When 0 is set, a retrace operation is performed at a programmed feedrate.

NOTE
This parameter has no effect on blocks that are in rapid traverse mode during reverse execution.

Warning

⚠ **WARNING**

1 Auxiliary functions are output directly even during reverse execution and forward reexecution. Accordingly, note that the execution status of an auxiliary function during forward execution may be reversed during reverse execution.
Example:
When forward rotation of the spindle(M03) and stop (M05) are specified.
When N3 is executed during reverse execution, M05 is output. So, when N2 and N1 are executed during reverse execution, operation is performed with the spindle stopped.
When N1 is executed during forward reexecution, M03 is output. So, when N1 and N2 are executed during forward reexecution, operation is performed with the spindle rotating in the forward direction.

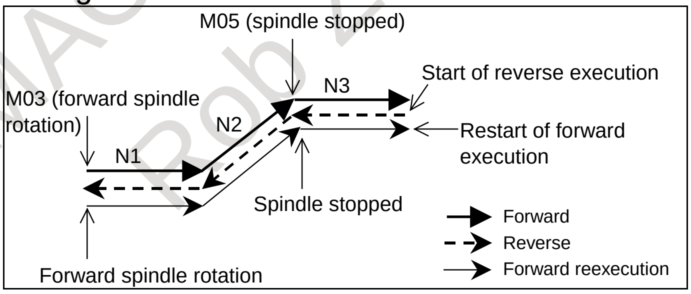


Fig. 5.10 (o)

2 To perform reverse execution after a feed hold stop or single block stop operation, be sure to restore the original position if manual intervention has been performed after the stop , then set the reverse execution signal to 1. Movements made by manual intervention are ignored during reverse execution and forward reexecution. (The same operation as in the manual absolute off state takes place.)
If manual intervention is performed during reverse execution or forward reexecution, the amount of manual intervention is added to the coordinate system at a restart after a stop due to a feed hold or single block during forward execution after the end of forward reexecution. Whether to add the amount of manual intervention follows the manual absolute signal *ABSM <Gn006.2>.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Retrace

5.11 EXACT STOP / EXACT STOP MODE / TAPPING MODE / CUTTING MODE

Overview

NC commands can be used to control a feedrate in continuous cutting feed blocks as described below.

- Exact stop (G09)

The tool is decelerated in a block specifying G09, and an in-position check (*1) is performed. When the feed motor falls in-position, the tool is moved by the next block. This function may be used to produce a sharp edge at the corner of a workpiece at cutting feed.

- Exact Stop Mode (G61)

When G61 is commanded, deceleration of cutting feed command at the end point and in-position check is performed per block thereafter. This G61 is valid until G62 (automatic corner override), G63 (tapping mode), or G64 (cutting mode), is commanded.

- Tapping Mode (G63)

When G63 is commanded, feed rate override is ignored (always regarded as 100%), and feed hold also becomes invalid. Cutting feed does not decelerate at the end of block to transfer to the next block. Tapping signal TAP<Fn001.5> is set to "1" during tapping mode. This G63 is valid until G61 (exact stop mode), G62 (automatic corner override), or G64 (cutting mode) is commanded.

> **NOTE**
> Spindle override is enabled during tapping mode.

- Cutting Mode (G64)

When G64 is commanded, deceleration at the end point of each cutting feed block thereafter is not performed and cutting goes on to the next block. This command is valid until G61 (exact stop mode), G62 (automatic corner override), or G63 (tapping mode) is commanded. However, in G64 mode, feed rate is decelerated to zero and in-position check is performed in the following case;
1) Positioning mode (G00, G60)
2) Block with exact stop check (G09)
3) Next block is a block without movement command

*1 The term in-position indicates that the servo motor reaches in a range of positions specified by a parameter. See Section "In-position check" and "In-position check independently of feed/rapid traverse" for details.

> **NOTE**
> When the in-position check signal SMZ<Gn053.6> is set to "1", cutting feed decelerates at the end point of each block and an in-position check is performed even in tapping mode or cutting mode.

(Example) Tool paths from block <1> to block <2>

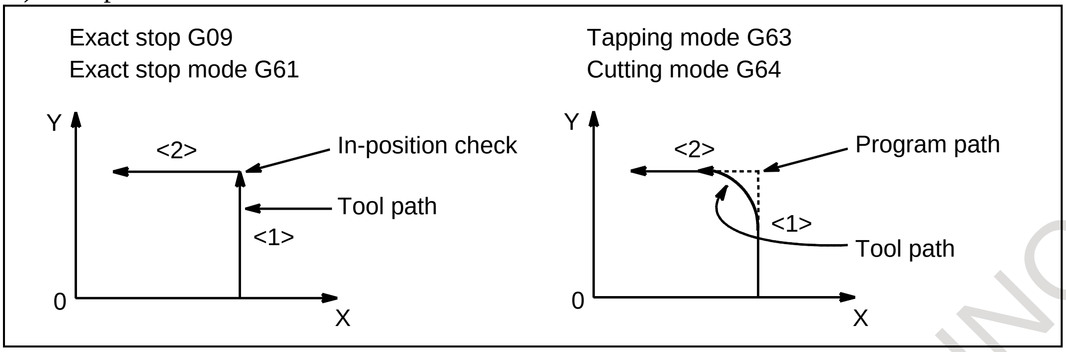


Fig. 5.11 (a)

- Restriction on tapping mode (G63)

There are restrictions when the following functions are used. Please refer to each function for detail.

- Retrace
- AI contour control

Signal

Tapping signal TAP<Fn001.5>

[Classification] Output signal

[Function] Reports that tapping is in progress.

[Output cond.] The signal is set to "1" when:

- The system is in tapping cycle mode.
  (G74,G84 : for M series)
  (G84,G88 : for T series)
- The system is in tapping mode.
  (G63)

The signal is set to "0" when:

- The system is in neither tapping cycle mode nor tapping mode.

> **NOTE**
> When a reset or emergency stop is specified in tapping cycle mode, if the system is not in tapping mode, this signal is set to "0".

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn001			TAP					

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Exact stop (G09,G61) Cutting mode (G64) Tapping mode (G63)

5.12 RETRACTION FOR RIGID TAPPING / RETRACTION FOR 3-DIMENSIONAL RIGID TAPPING

Overview

When rigid tapping is stopped by a result of a power shutdown, emergency stop, or reset, the tap may cut into the workpiece. The tap can subsequently be drawn out by using a PMC signal or a program command. This function automatically stores information on tapping executed most recently. When a tap retraction

signal is input or G30 is specified, only retraction in the rigid tapping cycle is executed, based on the stored information, and the tap is removed and pulled toward the R point. When a retract value α is set in parameter No. 5382, the retraction distance can be increased by α.
The machining data for rigid tapping retraction is retained until the next rigid tapping command is specified, even after the power is turned off. Rigid tapping retraction can, therefore, be specified even after the power is turned off.
In retraction for 3-dimensional rigid tapping, the tap is removed and pulled when 3-dimensional rigid tapping or rigid tapping in tilted working plane indexing command mode is stopped.

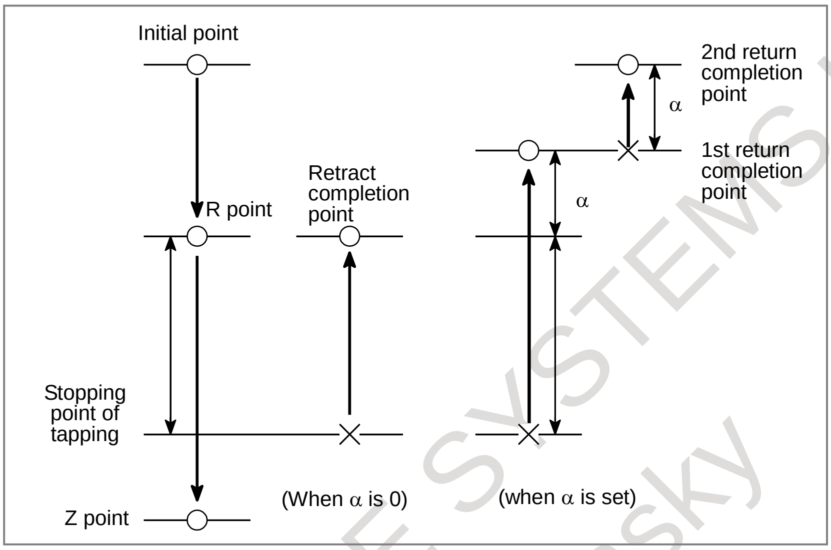


Fig. 5.12 (a)

> **NOTE**
> 1 Retraction for rigid tapping and 3-dimensional rigid tapping are optional functions.
> 2 To use retraction for rigid tapping, option of rigid tapping is also required.
> 3 To use 3-dimensional rigid tapping, option of retraction for rigid tapping is also required.

Basic procedure

- Retraction for rigid tapping by using a PMC signal

This method is used when bit 1 (RG3) of parameter No. 5202 is 0.

1 Start
Reset the CNC, then select MDI mode. Setting rigid tapping retraction start signal RTNT <Gn062.6> to “1” starts rigid tapping retraction.

2 Completion
Upon the completion of rigid tapping retraction, rigid tapping retraction completion signal RTPT <Fn066.1> is set to “1”, with which the CNC automatically enters the reset state. Setting rigid tapping retract start signal RTNT to “0” sets rigid tapping retraction completion signal RTPT to “0”.

3 Stop
During rigid tapping retraction, setting rigid tapping retraction start signal RTNT to “0” stops rigid tapping retraction, placing the CNC in the reset state. To resume rigid tapping retraction, set rigid tapping retraction start signal RTNT to “1”. Rigid tapping retraction can also be stopped by means of a reset or feed hold.

4 Resume
Once rigid tapping retraction has been stopped, it can be resumed by performing the same operation as that used for starting rigid tapping retraction. If rigid tapping retraction has been completed, however, the start operation does not restart rigid tapping retraction. If retract value α is set in parameter No. 5382, however, the start operation performs rigid tapping retraction using α only.

- Start and completion time chart

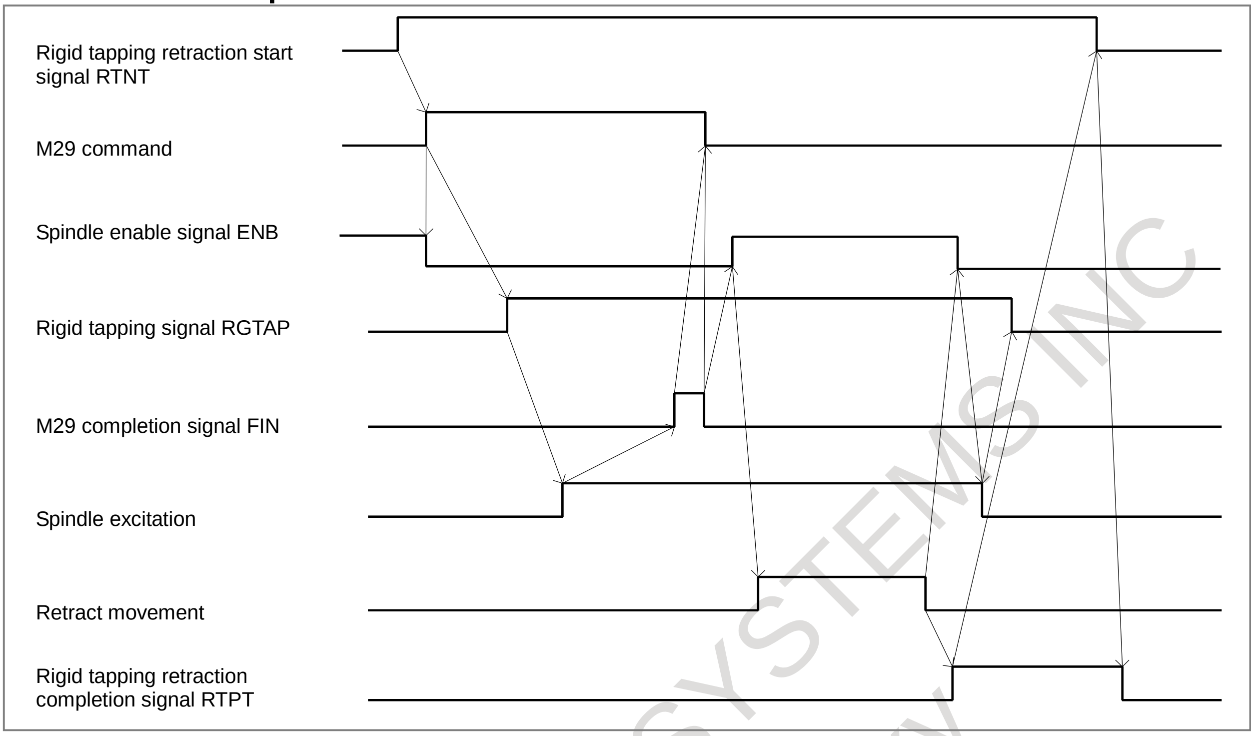


Fig. 5.12 (b) Start and completion time chart

In the reset state, setting rigid tapping retraction start signal RTNT to "1" in MDI mode causes the rigid tapping M command to be output. For rigid tapping retraction, specify neither gear switching nor orientation. Spindle function strobe signal SF is also output if no S command has been specified after power-on.
Upon the completion of rigid tapping retraction, spindle enable signal ENB <Fn001.4> is set to "0", in the same way as at the end of ordinary rigid tapping. Therefore, perform the sequence for canceling rigid tapping. Once rigid tapping retraction has been completed, rigid tapping retraction completion signal RTPT is set to 1 and the CNC enters the reset state.

- Time chart for stopping tapping retraction

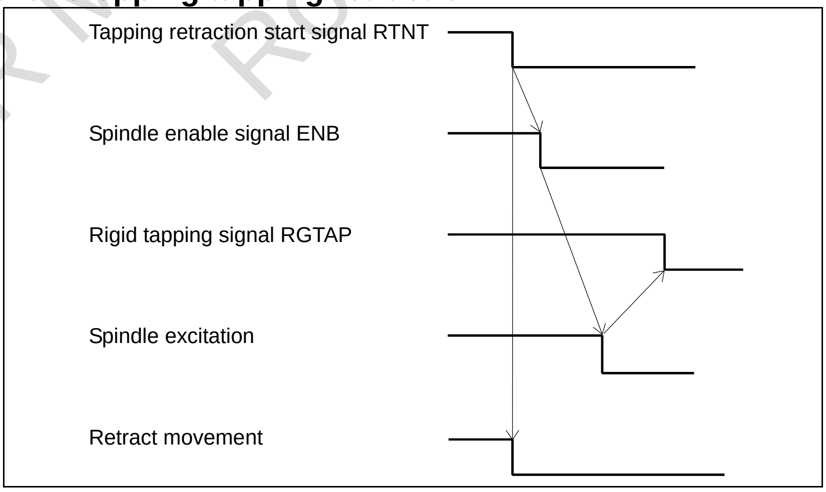


Fig. 5.12 (c)

When tapping retraction is stopped, spindle enable signal is set to "0", in the same way as for ordinary rigid tapping. Therefore, perform the sequence for canceling rigid tapping. The CNC also automatically enters the reset state when tapping retraction is stopped.

- Retraction for rigid tapping by using the G30 command

If bit 1 (RG3) of parameter No. 5202 is 1, it is possible to perform retraction for rigid tapping by using the G30 command.
If rigid tapping is interrupted due to a reset or emergency stop or if the power is turned off during rigid tapping, executing the command below in MEM or MDI operation causes the tapping axis to move to the initial point or the R point while synchronized with the spindle, based on the rigid tapping command information in the machining program.
The command for retraction for rigid tapping is a one shot command.

- Retraction for rigid tapping
  G30 P99 M29 S $\underline{\text{min}^{-1}}$;

(Example)

```
Machining program
----------
M29 S1000 ;
G84 X20.0 Y20.0 R-10.0 Z-30.0 F500 ;
X50.0 Y50.0 ;
X100.0 Y100.0 ;
G80
----------

Retraction program
----------
G30 P99 M29 S1000 ;
G00 Z-10.0 ;
----------
```

- Retraction for rigid tapping during 3-dimensional coordinate system conversion

G68 Xxx Yyy Zzz Iii Jjj Kkk Rrr ;	(3-dimensional coordinate system conversion) (NOTE 1)
G30 P99 M29 Sxxx ;	(Retraction for rigid tapping) (NOTE 2)

- Retraction for rigid tapping in tilted working plane indexing mode

G68.2 Xxx Yyy Zzz Iii Jjj Kkk Rrr ;	(Tilted working plane indexing) (NOTE 1)
G53.1 ;	(Tool axis direction control)
G30 P99 M29 Sxxx ;	(Retraction for rigid tapping) (NOTE 2)

NOTE
1. Specify arguments used when rigid tapping stops.Alarm PS5384 occurs if the specified arguments are different from those used at stopping.
2. M-code and S-code used when rigid tapping stops are executed regardless of the "M29 Sxxxx" command. "M29 Sxxxx" can be omitted from the G30 P99 command.
3. If a value other than 0 is set in parameter No. 5210, specify the value instead of M29 in the above program.
4. Be sure to set bit 0 (G84) of parameter No. 5200 to 0 before using this function.

Limitation

T

When G code system A is used for a lathe system, retraction for rigid tapping using Rigid tapping retraction start signal RTNT <Gn062.6> (parameter RG3(bit 1 of No.5202)=1), you may want to use an axis parallel to one of the basic three axes as a drilling axis. In this case, use the extended axis name function to specify X*, Y*, or Z* (* is an alphanumeric character) as the axis name of the axis parallel to one of the basic three axes.

Examples

Basic three axes: X, Y, and Z

Parallel axes: XA, YA, and ZA

Note

1. When retraction involves movement only for the retract value set in parameter No. 5382, the retraction operation is executed in the program coordinate system in 3-dimensional coordinate system conversion mode or tilted working plane indexing mode.
2. Retraction for rigid tapping by the G30 command results in alarm PS5384 if the program coordinate system used when rigid tapping is stopped is different from that used for retraction for rigid tapping. The program coordinate system is the coordinate system for the program after conversion is determined by the rotation center, the direction of rotation, and the rotation angle that was commanded by the arguments of the G68 or G68.2 command. In other words, the arguments of G68 or G68.2 command are different at the stop of rigid tapping and in retraction for rigid tapping.

Signal

Rigid tapping signal RGTAP <Gn061.0>

[Classification] Input signal

[Function] The M29 (rigid tapping mode ready auxiliary function) command causes the PMC to place itself in rigid tapping mode, turning this signal ON and notifying the CNC.

1 : The PMC is in rigid tapping mode.

0 : The PMC is not in rigid tapping mode.

For an explanation of the processing for placing the PMC in rigid tapping mode, see "Interface with the PMC", to be described later. This signal is a check signal used to determine whether the PMC is placed in rigid tapping mode. If M29 is specified, but this signal is not set to “1”, a PS alarm will be issued in a G84/G74 (machining center system) or G84/G88 (lathe system) block.

Rigid tapping retraction start signal RTNT <Gn062.6>

[Classification] Input signal

[Function] Starts rigid tapping retraction.

[Operation] When this signal is set to “1”, the control unit operates as follows:

- Starts rigid tapping retraction.

Rigid tapping retraction completion signal RTPT <Fn066.1>

[Classification] Output signal

[Function] Notifies the completion of rigid tapping retraction.

[Output cond.] This signal is set to “1” in the following case:

- Rigid tapping retraction has been completed.

This signal is set to “0” in the following case:

- Rigid tapping retraction start signal has been set to “0”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn061								RGTAP
Gn062		RTNT						
Fn066							RTPT	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5200				DOV				

[Input type] Parameter input
[Data type] Bit path

#4 DOV Override during extraction in rigid tapping:
0: Invalidated
1: Validated (The override value is set in parameter No. 5211. However, set an override value for rigid tapping return in parameter No. 5381.)

	#7	#6	#5	#4	#3	#2	#1	#0
5201					OVU			

[Input type] Parameter input
[Data type] Bit path

#3 OVU The increment unit of the override parameter No. 5211 for tool rigid tapping extraction and No. 5381 for tool rigid tapping return is:
0: 1%
1: 10%

	#7	#6	#5	#4	#3	#2	#1	#0
5202							RG3	

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 RG3 Retraction for rigid tapping is performed by:
0: Rigid tapping retraction start signal RTNT <Gn062.6>.
1: One-shot G code G30 command.

> **NOTE**
> 1 When this parameter is 1, retraction for rigid tapping using the rigid tapping retraction start signal RTNT <Gn062.6> cannot be performed.
> 2 When this parameter is 1, use the method to use the M code of rigid tapping mode. (Parameter G84(bit 0 of No.5200)=0)

5381	Override value during rigid tapping return

[Input type] Parameter input
[Data type] Word path
[Unit of data] % or 10% (The unit of this parameter depends on the parameters OVU (bit 3 of parameter No.5201))
[Valid data range] 0 to 200
This parameter is used to set the override value during rigid tapping return.
If the setting is 0, no override is applied.

NOTE
1. This parameter is valid when bit 4 (DOV) of parameter No. 5200 for enabling override at normal extraction time is set to 1.
2. When the parameter OVU (bit 3 of No.5201) is set to 1, the data unit of this parameter is 10%.
3. If the setting value is less than or equal to 0, it is treated as 100%. Also, if the setting value is greater than 200, it is treated as 2000% or 200%.

5382	Amount of return for rigid tapping return

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the drilling axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

This parameter is used to set an extra amount of rigid tapping return. The tool is retracted additionally near point R by the distance set in this parameter. If the tool has already been retracted from rigid tapping, it will be retracted further only by the distance specified in this parameter.

NOTE
If a negative value is set in this parameter, it is treated as a positive value.

Warning

⚠ WARNING
1. For rigid tapping retraction, the CNC internally activates a return program. Rigid tapping retraction may, therefore, cause some G codes or M/F/S/L codes to be overwritten (G80/G84/G74, G94/G95).
2. During rigid tapping retraction, do not change any setting affecting the travel distance (such as the increment system, input unit 10 time multiply, or calculator-type decimal notation) that was made when machining data was stored for rigid tapping retraction.

Caution

⚠ CAUTION
1. If rigid tapping is stopped as a result of an emergency stop, the position on the tapping axis (Z-axis) is maintained but the spindle position is lost. In such a case, therefore, the positional relationship between the spindle and tapping axis is not guaranteed when operation is resumed.
2. Rigid tapping retraction is performed based on the tapping axis (Z-axis) commands accumulated for tapping. If rigid tapping is stopped as a result of an emergency stop, therefore, rigid tapping retraction may fail to draw the tapping tool completely out of the workpiece. In such a case, set retract value α (parameter No. 5382).
3. During rigid tapping retraction, switching the mode to the manual operation mode stops rigid tapping retraction, and decelerates and stops the machine.

> ⚠ **CAUTION**
> 4 If the power is turned off during rigid tapping motion by power failure or etc, when retraction for rigid tapping is executed after the power turned on, the tapping axis (Z-axis) may be retracted additionally by the accumulated distance of acceleration/deceleration.

Note

> **NOTE**
> 1 Setting rigid tapping retraction start signal RTNT <Gn062.6> to “1” starts rigid tapping retraction only when the CNC is placed in both the reset state and MDI mode.
> 2 Rigid tapping retraction is not performed if the input increment (inches or millimeters) selected when machining data is stored for rigid tapping retraction differs from that selected when rigid tapping retraction is executed.
> 3 Override can be applied to rigid tapping retraction when bit 4 (DOV) of parameter No. 5200 is set to 1.
> 4 Set each parameter before executing retract for rigid tapping.

Alarm and Message

Number	Message	Description
PS5384	RETRACT FOR RIGID CANNOT BE CMD.	In retraction for 3-dimensional rigid tapping by the G30 command, program coordinate used when rigid tapping is stopped and that used for retraction for rigid tapping are different. The program coordinate system is the coordinate system for the program after conversion is determined by the rotation center, the direction of rotation, and the rotation angle that was commanded by the arguments of the G68 or G68.2 command. In other words, the arguments of G68 or G68.2 command are different at the stop of rigid tapping and in retraction for rigid tapping. Modify the program.

5.13 DNC OPERATION

Overview

When automatic operation is performed in the DNC operation mode (RMT), machining (DNC operation) can be performed while a program is being read via the RS-232C interface, from the Embedded Ethernet or from a memory card.
Before performing DNC operation, set parameters relating to the RS-232C interface.

> **NOTE**
> DNC operation cannot be performed while a program is being read from a USB memory.

Signal

DNC operation select signal DNCI<Gn043.5>

[Classification] Input signal

[Function] Selects the DNC operation mode (RMT).
To select the DNC operation mode (RMT), the memory operation mode (MEM) must be selected, and the DNC operation selection signal must be set to “1” at the same time.

[Operation] When this signal is set to “1”, the control unit operates as follows:
- If the memory operation mode (MEM) is not selected, the control unit ignores this signal and does nothing.
- If the memory operation mode (MEM) is selected, the DNC operation mode (RMT) is selected, enabling DNC operation. At this time, DNC operation selection confirm signal MRMT is set to “1”.

DNC operation selection confirm signal MRMT<Fn003.4>

[Classification] Output signal

[Function] Indicates that the DNC operation mode (RMT) is selected.

[Output cond.] This signal is set to “1” when:
- The DNC operation mode (RMT) is selected.

This signal is set to “0” when:
- Selection of the DNC operation mode (RMT) ends.

External device program execution signal DVCPR<Fn531.6>

[Classification] Output signal

[Function] The external device program is notified to being executed.

[Output cond.] This signal is set to “1” when:
- DNC operation
- M198 is being executed in MEM/MDI mode.

This signal is set to “0” when:
- Reset
- Switching to other mode (MEM/MDI) while DNC mode.
- Switching to other mode (MDI/DNC) while executing M198 in MEM mode.
- Switching to other mode (MEM/DNC) while executing M198 in MDI mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn043			DNCI					
Fn003				MRMT				
Fn531		DVCPR						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0138	MNC							

[Input type] Parameter input
[Data type] Bit

#7 MNC DNC operation from the memory card and external device subprogram call from the memory card are:
0: Not performed.
1: Performed. (In addition, compact flash card adapter is necessary)

NOTE
1 The compact flash card must be inserted in the front PCMCIA slot by using a compact flash card adapter.
2 When a DNC operation using a memory card is being performed, the memory card cannot be accessed to, for example, display a list of data stored on the memory card.
3 DNC operation using a memory card cannot be performed when the system has multiple paths.
4 During DNC operation using a memory card, never remove the memory card.
5 A program being executed by DNC operation cannot call another program on the memory card.
6 The memory card utilities of the CNC screen display function cannot be used to perform DNC operation.

	#7	#6	#5	#4	#3	#2	#1	#0
14885E							EDE	ISO

[Input type] Parameter input
[Data type] Bit

#0 ISO When the embedded Ethernet is selected as an I/O device, data input/output is performed using
0: ASCII codes.
1: ISO codes.

⚠ **WARNING**
1 Unless data is input using ASCII codes, set this parameter to 1 to input or output data using ISO codes.
2 Data input/output with ASCII codes is dangerous because parity information is not included and a data error during the data input/output is not detected.
3 DNC operation from the embedded Ethernet also must set the parameter to 1, and execute DNC operation by ISO code. Data input/output with ASCII codes is dangerous because parity information is not included and a data error during the data input is not detected.

#1 EDE DNC operation from the embedded Ethernet and external device subprogram call from the embedded Ethernet are:
0: Not performed.
1: performed.

NOTE
1 When DNC operation, check that the valid device is the embedded Ethernet port.
2 When a DNC operation using the embedded Ethernet is being performed, the embedded Ethernet cannot be accessed to, for example, display the Embedded Ethernet host file list screen.
3 DNC operation cannot be performed concurrently on multiple paths. DNC operation can be performed on one path at a time.
4 Only one DNC operation file can be set as a system.

Alarm and message

Number	Message	Description
SR0086	DR OFF	During I/O process by RS232C interface 1, the data set ready input signal of the I/O device (DR) was 0. Possible causes are an I/O device not turn on, a broken cable, and a defective printed circuit board.
SR1830	DR OFF(2)	During I/O process by RS232C interface 2, the data set ready input signal of the I/O device (DR) was 0. Possible causes are an I/O device not turn on, a broken cable, and a defective printed circuit board.
PS0123	ILLEGAL MODE FOR GOTO/WHILE/ DO	A GOTO statement or WHILE–DO statement was found in the program in the MDI or DNC mode.
PS1081	EXT DEVICE SUB PROGRAM CALL MODE ERROR	The external device subprogram call is not possible in this mode.

5.14 DIRECT OPERATION BY PERSONAL COMPUTER FUNCTION

Overview

When automatic operation is started in the DNC operation mode (RMT), it becomes possible to perform machining (direct operation = DNC operation) while a program is being read from a personal computer function.

DNC operation

To perform direct operation using a personal computer function, the following settings are required:

1. Determine connection destination 1 or 2 according to the connection method as follows:

Connection method	Destination
First HSSB board inserted in an option slot of the LCD-mounted type FS0*i*-F (except the 15-/19-inch LCD type)	Destination 1
Stand-alone type FS0*i*-F (For the stand-alone type, connect the HSSB cable to COP21A.)	Destination 1
Second HSSB board inserted in an option slot of the LCD-mounted type FS0*i*-F	Destination 2
HSSB board inserted in an option slot of the Stand-alone type FS0*i*-F	Destination 2
First HSSB board inserted in an option slot of the LCD-mounted type FS0*i*-F (15-/19-inch LCD type)	Destination 2

2. Make settings according to the connection destination as listed below. There are two setting methods for connection destination 1.

Destination	Setting
Destination 1	DMMC<G0042.7>=1
	I/O CHANNEL (parameter No.0020))=15 DNCI<G043.5>=1
Destination 2	I/O CHANNEL (parameter No.0020))=16 DNCI<G043.5>=1

Signal

Direct operation select signal DMMC<Gn042.7>

[Classification] Input signal

[Function] Selects the mode (direct operation mode) for performing machining while reading a program from the personal computer function.

[Operation] When this signal is set to "1", the control unit operates as follows:

- If the memory operation mode (MEM) is not selected, the control unit ignores this signal and does nothing.
- If the memory operation mode (MEM) is selected, the direct operation mode is selected, enabling direct operation.

DNC operation select signal DNCI<Gn043.5>

[Classification] Input signal

[Function] Selects the DNC operation mode (RMT).
To select the DNC operation mode (RMT), the memory operation mode (MEM) must be selected, and the DNC operation selection signal must be set to "1" at the same time.

[Operation] When this signal is set to "1", the control unit operates as follows:

- If the memory operation mode (MEM) is not selected, the control unit ignores this signal and does nothing.
- If the memory operation mode (MEM) is selected, the DNC operation mode (RMT) is selected, enabling DNC operation. At this time, DNC operation selection confirm signal MRMT is set to "1".

DNC operation selection confirmation signal MRMT<Fn003.4>

[Classification] Output signal

[Function] Indicates that the DNC operation mode (RMT) is selected.

[Output cond.] This signal is set to "1" when:

- The DNC operation mode (RMT) is selected.

This signal is set to "0" when:

- Selection of the DNC operation mode (RMT) ends.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn042	DMMC							
Gn043			DNCI					
Fn003				MRMT				

5.15 DIRECT OPERATION BY C LANGUAGE EXECUTOR

Overview

When the DNC operation mode (RMT) is set, and automatic operation is started, it becomes possible to perform machining (direct operation = DNC operation) while reading a program from the C Language Executor.

Signal

Direct operation select signal DMMC<Gn042.7>

[Classification] Input signal

[Function] Selects the mode (direct operation mode) for performing machining while reading a program from the C Language Executor.

[Operation] When this signal is set to “1”, the control unit operates as follows:

- If the memory operation mode (MEM) is not selected, the control unit ignores this signal and does nothing.
- If the memory operation mode (MEM) is selected, the direct operation mode is selected, enabling direct operation.

DNC operation selection confirm signal MRMT<Fn003.4>

[Classification] Output signal

[Function] Posts that the DNC operation mode (RMT) is selected.

[Output cond.] This signal is set to “1” when:

- The DNC operation mode (RMT) is selected.

This signal is set to “0” when:

- Selection of the DNC operation mode (RMT) ends.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn042	DMMC							
Fn003				MRMT				

NOTE

As direct operation select signal DMMC<Gn042.7>, the same signal as the signal used for direct operation by personal computer function is used. Direct operation by personal computer function and direct operation by C Language Executor cannot be performed at the same time. If an attempt is made to use these functions at the same time, an application that executes the cnc_dncstart2 function first becomes valid.

6 INTERPOLATION FUNCTION

6.1 POSITIONING

Overview

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate. In the absolute command, coordinate value of the end point is programmed. In the incremental command the distance the tool moves is programmed.
The tool path is determined by selecting one of the following with bit 1 (LRP) of parameter No. 1401:

- Linear interpolation type positioning
  The tool is positioned using a straight path and a speed that is not higher than the rapid traverse of each axis but that assures the shortest positioning time. By changing the acceleration/ deceleration type from the constant acceleration/deceleration (inclination) type to the constant time (time constant) type with bit 4 (PRT) of parameter No. 1603, the tool can be moved along a specified path.
- Non-linear interpolation type positioning
  Positioning is performed with each axis independently at the rapid traverse rate. Generally, the tool path is not a straight line.

The rapid traverse rate in the G00 command is set to the parameter No.1420 for each axis independently by the machine tool builder. In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in-position.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401							LRP	

[Input type] Parameter input
[Data type] Bit path

#1 LRP Positioning (G00)
0: Positioning is performed with non-linear type positioning so that the tool moves along each axis independently at rapid traverse.
1: Positioning is performed with linear interpolation so that the tool moves in a straight line.
When using 3-dimensional coordinate system conversion, set this parameter to 1.

1420	Rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1603				PRT				

[Input type] Parameter input
[Data type] Bit path

#4 PRT For positioning of linear interpolation type:
0: Acceleration/deceleration of acceleration fixed type is used.
1: Acceleration/deceleration of time fixed type is used.

Note

NOTE
1. The rapid traverse rate cannot be specified in the address F.
2. In case of linear interpolation type positioning, set the parameters Nos. 1620 and 1621 to the same values for all axes. If the setting values are different in each axis, the positioning may not be straight path.
   If linear rapid traverse is acceleration/deceleration before interpolation (FRP(bit 5 of parameter No.19501) is set to 1), the positioning path is straight even if the setting values are different in each axes.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Positioning (G00)

6.2 SINGLE DIRECTION POSITIONING

Overview

For accurate positioning without play of the machine (backlash), final positioning from one direction is available.

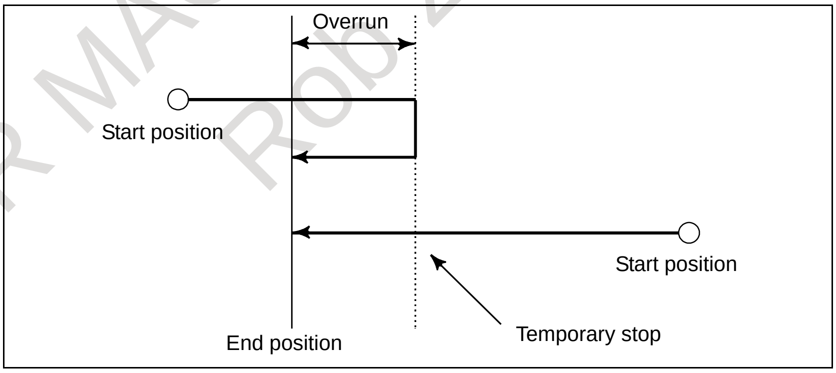


Fig. 6.2 (a) Example in which the positioning direction is the minus direction

An overrun and a positioning direction are set by the parameter No. 5440.
Even when a commanded positioning direction coincides with that set by the parameter, the tool stops once before the end point.
G60, which is an one-shot G-code, can be used as a modal G-code in group 01 by setting 1 to the bit 0 (MDL) of parameter No. 5431.
This setting can eliminate specifying a G60 command for every block. Other specifications are the same as those for an one-shot G60 command.
When an one-shot G code is specified in the single direction positioning mode, the one-shot G command is effective.

(Example)

When one-shot G60 commands are used.		When modal G60 command is used.	
:		:	
:		:	
G90;		G90 G60;	Single direction positioning mode start
G60 X0 Y0;	Single direction positioning	X0 Y0;	Single direction positioning
G60 X100;		X100;	
G60 Y100;		Y100;	
G04 X10;		G04 X10;	
G00 X0 Y0;		G00 X0 Y0;	Single direction positioning mode cancel

- Overview of operation

- In the case of positioning of non-linear interpolation type (bit 1 (LRP) of parameter No. 1401 = 0)

As shown Fig. 6.2(b), single direction positioning is performed independently along each axis.

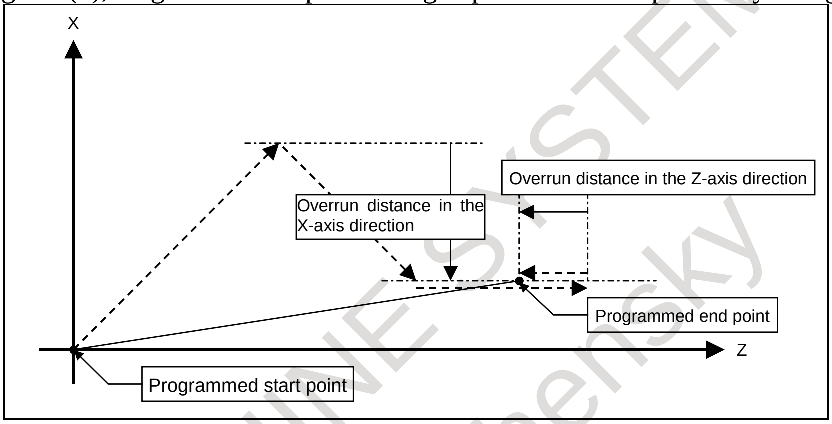


Fig. 6.2 (b)

- In the case of positioning of linear interpolation type (bit 1 (LRP) of parameter No. 1401 = 1)

Positioning of interpolation type is performed until the tool once stops before or after a specified end point. Then, the tool is positioned independently along each axis until the end point is reached.

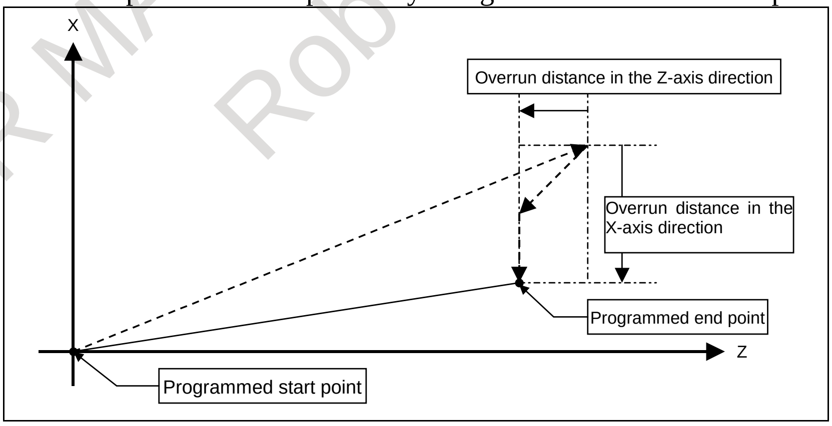


Fig. 6.2 (c)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5431							PDI	MDL

[Input type] Parameter input
[Data type] Bit

#0 MDL The G60 code (single direction positioning) is:
0: One-shot G code (group 00).
1: Modal G code (group 01).

#1 PDI In the G60 mode, an in-position check at a stop position is:
0: Not made. (Waiting for only the end of acceleration/deceleration)
1: Made.

5440	Positioning direction and overrun distance in single direction positioning

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (A)
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the positioning direction and overrun distance in single direction positioning (G60) for each axis. The positioning direction is specified using a setting data sign, and the overrun distance using a value set here.
Overrun distance>0: The positioning direction is positive (+).
Overrun distance<0: The positioning direction is negative (-).
Overrun distance=0: Single direction positioning is not performed.

Alarm And message

Number	Message	Contents
DS0025	G60 CANNOT BE EXECUTED	The state of mirror image when a single direction positioning block is looked ahead differs from the state of mirror image when the execution of the block is started, so that single direction positioning cannot be executed. Correct the program.

Limitation

- Single direction positioning is not performed along an axis for which no overrun distance is set in parameter No. 5440.
- Single direction positioning is not performed along an axis for which travel distance 0 is specified.
- The mirror image function is not applied in a parameter-set direction. Even in the mirror image mode, the direction of single direction positioning remains unchanged. If positioning of linear interpolation type is used, and the state of mirror image when a single direction positioning block is looked ahead differs from the state of mirror image when the execution of the block is started, an alarm (DS0025)"G60 CANNOT BE EXECUTED" is issued. When switching mirror image in the middle of a program, disable looking ahead by specifying a non-buffering M code. Then, switch mirror image when there is no look-ahead block.
- In the cylindrical interpolation mode (G07.1), single direction positioning cannot be used.
- In the polar coordinate interpolation mode (G12.1), single direction positioning cannot be used.

- When specifying single direction positioning on a machine that uses angular axis control, first position the angular axis then specify the positioning of the Cartesian axis. If the reverse specification order is used, or the angular axis and Cartesian axis are specified in the same block, an incorrect positioning direction can result.
- In positioning at a restart position by program restart function, single direction positioning is not performed.

M

- Single direction positioning cannot be performed for a hole machining axis in a canned cycle for hole machining.
- Single direction positioning is not performed for an axis subject to shift value movement in a G76 or G87 canned cycle.

T

- The G code for single direction positioning is G60, regardless of whether the G code system is A, B, or C.
- In a multiple repetitive canned cycle (G70 to G76), single direction positioning cannot be used.
- Single direction positioning cannot be used for a hole machining axis in a canned cycle for hole machining (G83 to G89) and rigid tapping (G84 and G88). However, single direction positioning can be used for positioning operation.
- Single direction positioning cannot be used in a canned cycle (G90, G92, or G94).
- In the single direction positioning mode, G07.1, G12.1, G70 to G76, and G90 to G94 cannot be specified.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Single direction positioning

6.3 LINEAR INTERPOLATION

Overview

Tools can move along a line. A tools move along a line to the specified position at the feedrate specified in F.

The feedrate specified in F is effective until a new value is specified. It need not be specified for each block.

The feedrate commanded by the F code is measured along the tool path. If the F code is not commanded, the feedrate is regarded as zero.

The feedrate of each axis direction is as follows.

G01 $\alpha\underline{\alpha}$ $\beta\underline{\beta}$ $\gamma\underline{\gamma}$ $\zeta\underline{\zeta}$;

Feed rate of α axis direction : $F\alpha=\alpha/L\times f$

Feed rate of β axis direction : $F\beta=\beta/L\times f$

Feed rate of γ axis direction : $F\gamma=\gamma/L\times f$

Feed rate of ζ axis direction : $F\zeta=\zeta/L\times f$

$$L=\sqrt{\alpha^2+\beta^2+\gamma^2+\zeta^2}$$

The feedrate of the rotary axis is commanded in the unit of deg/min (if the feedrate is 12 deg/min, F12.0 is commanded).

When the straight line axis α (such as X, Y, or Z) and the rotating axis β (such as A, B, or C) are linearly interpolated, the feed rate is that in which the tangential feed rate in the α and β cartesian coordinate system is commanded by F(mm/min).

β-axis feedrate is obtained ; at first, the time required for distribution is calculated by using the above formula, then the β-axis feedrate unit is changed to deg/min.
A calculation example is as follows.
(Example)

G91 G01 X20.0 C40.0 F300.0 ;
This changes the unit of the C axis from 40.0 deg to 40mm with metric input. The time required for distribution is calculated as follows:

$$\frac{\sqrt{20^2+40^2}}{300} \fallingdotseq 0.14907\ \mathrm{min}$$

The feed rate for the C axis is

$$\frac{40\ \mathrm{deg}}{0.14907\ \mathrm{min}} \fallingdotseq 268.3\mathrm{deg/min}$$

In simultaneous 3 axes control, the feed rate is calculated in cartesian coordinate system the same way as in simultaneous 2 axes control.

Parameter

1411	Cutting feedrate

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
A cutting feedrate can be specified with this parameter for a machine which does not have to change the cutting feedrate frequently during machining. This eliminates the need to specify a cutting feedrate (F code) in the NC program.

1430	Maximum cutting feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Specify the maximum cutting feedrate for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
3402								G01

[Input type] Parameter input
[Data type] Bit path

#0 G01 G01 Mode entered when the power is turned on or when the control is cleared
0: G00 mode (positioning)
1: G01 mode (linear interpolation)

Alarm and message

Number	Message	Description
PS0011	FEED ZERO (COMMAND)	The cutting feedrate instructed by an F code has been set to 0. This alarm is also generated if the F code instructed for the S code is set extremely small in a rigid tapping instruction as the tool cannot cut at the programmed lead.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Linear interpolation (G01)

6.4 CIRCULAR INTERPOLATION

Overview

The command below can move a tool along a circular arc in the defined plane.
"Clockwise"(G02) and "counterclockwise"(G03) on the XpYp plane (ZpXp plane or YpZp plane) are defined when the XpYp plane is viewed in the positive-to-negative direction of the Zp axis (Yp axis or Xp axis, respectively) in the right-handed Cartesian coordinate system. See the Fig. 6.4 (a).

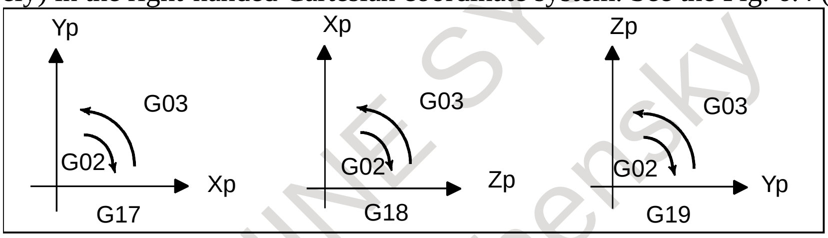


Fig. 6.4 (a)

The end point of an arc is specified by address Xp, Yp or Zp, and is expressed as an absolute or incremental value according to G90 or G91. For the incremental value, the distance of the end point which is viewed from the start point of the arc is specified with a sign.
The arc center is specified by addresses I, J, and K for the Xp, Yp, and Zp axes, respectively. The numerical value following I, J, or K, however, is a vector component in which the arc center is seen from the start point, and is always specified as an incremental value, as shown below (Fig. 6.4 (b)).
I, J, and K must be signed according to the direction.

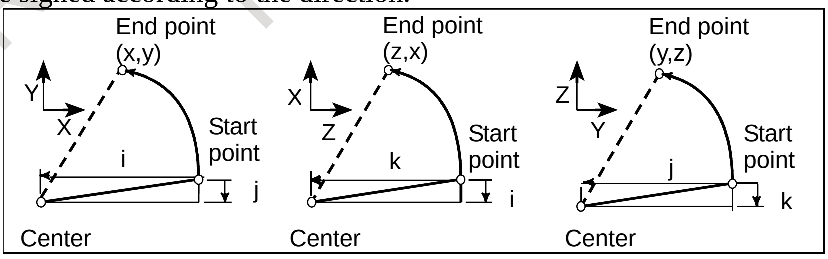


Fig. 6.4 (b)

I0,J0, and K0 can be omitted. When Xp, Yp , and Zp are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc (circle) is specified.
G02 I; Command for a circle
If the difference between the radius at the start point and that at the end point exceeds the value in a parameter No. 3410, an alarm PS0020 occurs.
The distance between an arc and the center of a circle that contains the arc can be specified using the radius, R, of the circle instead of I, J, and K.
In this case, one arc is less than 180°, and the other is more than 180° are considered.

Specify an arc more than 180° with a negative radius value commanded.
If Xp, Yp, and Zp are all omitted, if the end point is located at the same position as the start point and when R is used, an arc of 05 is programmed.
G02R_ ; (The tool does not move.)

(Example)
For arc (1) (less than 180°)
G91 G02 X60.0 Y20.0 R50.0 F300.0 ;
For arc (2) (greater than 180°)
G91 G02 X60.0 Y20.0 R-50.0 F300.0 ;

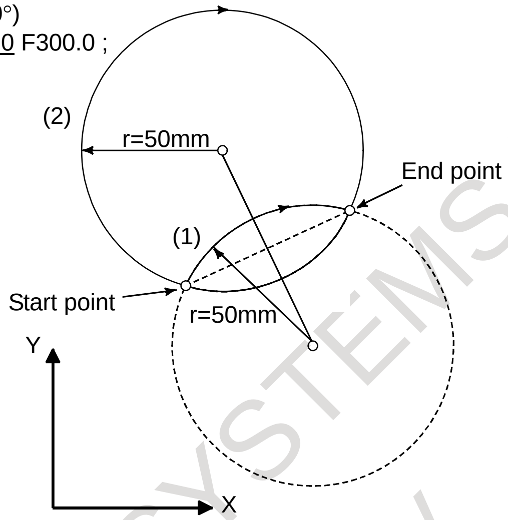


Fig. 6.4 (c)

The feedrate in circular interpolation is equal to the feedrate specified by the F code, and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate.

Parameter

1022	Setting of each axis in the basic coordinate system

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

To determine a plane for circular interpolation, cutter compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane), specify which of the basic three axes (X, Y, and Z) is used for each control axis, or a parallel axis of which basic axis is used for each control axis.
A basic axis (X, Y, or Z) can be specified only for one control axis.
Two or more control axes can be set as parallel axes for the same basic axis.

Table 6.4 (a)

Setting	Meaning
0	Rotary axis (Neither the basic three axes nor a parallel axis)
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

In general, the increment system and diameter/radius specification of an axis set as a parallel axis are to be set in the same way as for the basic three axes.

	#7	#6	#5	#4	#3	#2	#1	#0
3402								
						G19	G18	

[Input type] Parameter input
[Data type] Bit path

#1 G18 Plane selected when power is turned on or when the control is cleared
0: G17 mode (plane XY)
1: G18 mode (plane ZX)

#2 G19 Plane selected when power is turned on or when the control is cleared
0: The setting of bit 1 (G18) of parameter No. 3402 is followed.
1: G19 mode (plane YZ)
When this bit is set to 1, set bit 1 (G18) of parameter No. 3402 to 0.

3410	Tolerance of arc radius

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
When a circular interpolation command is executed, the tolerance for the radius between the start point and the end point is set.

Alarm and message

Number	Message	Description
PS0011	FEED ZERO (COMMAND)	The cutting feedrate instructed by an F code has been set to 0. This alarm is also generated if the F code instructed for the S code is set extremely small in a rigid tapping instruction as the tool cannot cut at the programmed lead.
PS0020	OVER TOLERANCE OF RADIUS	An arc was specified for which the difference in the radius at the start and end points exceeds the value set in parameter No. 3410. Check arc center codes I, J and K in the program. The tool path when parameter No. 3410 is set to a large value is spiral.
PS0021	ILLEGAL PLANE SELECT	1) In the selected plane (by G17, G18 or G19 command), the basic axis(X,Y, or Z) and its parallel axis are commanded at the same time. Modify the program. 2) In the following functions, an axis not included in the selected plane was commanded,or the selected plane was not correct. Modify the program : - Circular interpolation - Multiple repetitive cycle (if parameter DSA(No.5109#0) is set to 1.) etc. 3) In the helical interpolation, the number of axes that can be commanded simultaneously is exceeded. Modify the program.
PS0025	CIRCLE CUT IN RAPID (F0)	F0 (rapid traverse in inverse feed or feed specified by an F code with 1–digit number) was specified during circular interpolation (G02, G03).

Note

NOTE

1. For T series, the U, V and W axes (parallel with the basic axis) can be used with G-code system B and C.
2. If I, J, K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.
3. If an axis not comprising the specified plane is commanded, an alarm is displayed.
   For example, when G code system B or C is used, if U axis with X axis is specified as a parallel axis to X axis when plane XY is specified, an alarm PS0021 is displayed.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Circular interpolation (G02, G03)

6.5 THREADING

6.5.1 Threading

Overview

Tool movement can be synchronized with spindle rotation when cutting threads.
The spindle speed is continuously read through the position coder attached to the spindle. Then, it is converted to a cutting feedrate (feed per minute) to feed the tool.

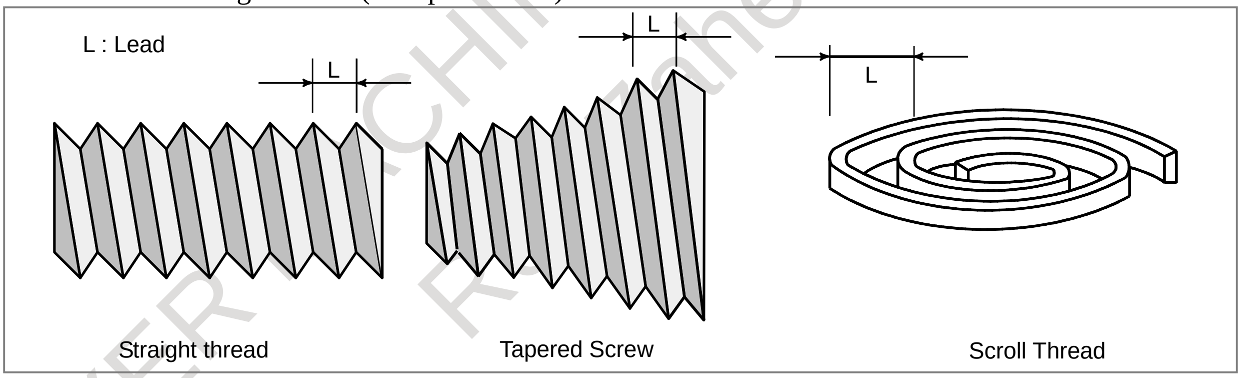


Fig. 6.5.1 (a)

In general, thread cutting is repeated along the same tool path in rough cutting through finish cutting for a screw. Since thread cutting starts when the position coder mounted on the spindle outputs a 1-turn signal, threading is started at a fixed point and the tool path on the workpiece is unchanged for repeated thread cutting. Note that the spindle speed must remain constant from rough cutting through finish cutting. If not, incorrect thread lead will occur.

⚠ **CAUTION**

Detector with one-rotation signal is required for threading, and it must be connected with spindle at a gear ratio of 1:1.
If detector is not connected correctly, the lead of threading may become wrong or start position of threading may become wrong.

- Serial spindle

Set the parameters correctly according to detector configuration. For detector configuration and the parameters, refer to FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN).

- Analog spindle

Connect position coder with 4096 pulses per one rotation of the spindle.

It is enabled that feedback pulses of serial spindle or analog spindle is changed to high precision. For details, refer to "11.27 IMPROVEMENT OF SPINDLE FEEDBACK PULSES".

- When the maximum cutting feedrate is exceeded

If bit 4 (MTA) of parameter No.1612 is 0 and the maximum cutting feedrate (parameter No.1430) is exceeded, the alarm PS0530 “EXCESS VELOCITY IN THREADING” is issued as follows:

- When the maximum cutting feedrate is exceeded before threading start, the alarm is issued and the tool stops immediately.
- When the maximum cutting feedrate is exceeded after threading start, the alarm is issued and the tool stops after threading is finished.

This function can detect that the maximum cutting feedrate is exceeded. If the bit 4 (MTA) of parameter No.1612 is 0, the alarm is not issued.

NOTE
The alarm PS0530 is not issued during dry run.

Limitation

- Tool Retract and Recover

When the major axis for threading is specified as the retraction axis, retraction is not performed. In this case, after a block that does not specify threading is executed, an alarm PS0429, “ILLEGAL COMMAND IN G10.6” is issued and the tool stops.

T

- Chamfering and Corner R

Chamfering or corner R cannot be specified in a block in which thread cutting is specified. If chamfering or corner R is specified, alarm PS0050, “CHF/CNR NOT ALLOWED IN THRD BLK”, is issued.

- Tool offset

Tool offset cannot be specified in a block in which thread cutting is specified. If tool offset is specified, alarm PS0509, “TOOL OFFSET COMMAND IS NOT AVAILABLE”, is issued.

Signal

Spindle speed arrival signal SAR<Gn029.4>

[Classification] Input signal

[Function] The SAR signal initiates cutting feed. In other words, if the signal is logical “0”, cutting feed will not start.

[Operation] Generally, this signal is used to inform the CNC that the spindle has reached the specified speed. For this purpose, the signal must be set to “1” only after the actual speed of the spindle has reached the specified speed.
Setting parameter No. 3740 with a wait time before the start of checking the SAR signal inhibits cutting feed from starting under a condition of SAR = “1” specified before the change of the spindle command.
To use the SAR signal, it is necessary to set bit 0 (SAR) of parameter No. 3708 to 1. The CNC checks the SAR signal under the following conditions:
a. Bit 0 (SAR) of parameter No. 3708 is set to 1.

b. Before starting distribution of the first feed (move command) block after shifting from the rapid traverse mode to the cutting feed mode. This checking is performed after the time set by parameter No. 3740 has elapsed after the feed block is read.
c. Before starting distribution of the first feed command block after an S code is commanded. The wait time for checking is the same as in item b.
d. When an S code and feed are programmed in the same block, the S code (or command output to the spindle) is output, and the SAR signal is checked after a fixed time elapses. If the SAR signal is set to "1", feed begins.

> ⚠ **CAUTION**
> According to the conditions of item d above, note that if the circuit is so designed that SAR is turned to "0" simultaneously with the output of an S code and the change of spindle speed is initiated by the DEN signal, the operation will stop. That is, the spindle speed does not reach the commanded speed because the CNC is waiting for the DEN signal and distribution is not started because the CNC is waiting for the SAR signal.

Threading signal THRD<Fn002.3>

[Function] This signal indicates that threading is in progress.

[Output cond.] This signal turns to "1" in the following cases:
- Threading mode in progress
- Threading cycle for turning

This signal turns to "0" in the following case.
- Neither threading mode nor thread cutting cycle are in progress.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn029				SAR				

	#7	#6	#5	#4	#3	#2	#1	#0
Fn002					THRD			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401			TDR					

[Input type] Parameter input
[Data type] Bit path

#5 TDR Dry run during threading or tapping (tapping cycle G74 or G84, rigid tapping)
0: Enabled
1: Disabled

	#7	#6	#5	#4	#3	#2	#1	#0
1402				JRV				NPC

[Input type] Parameter input
[Data type] Bit path

#0 NPC Feed per revolution without the position coder (function for converting feed per revolution F to feed per minute F in the feed per revolution mode (G95)) is:
0: Not used
1: Used

#4 JRV Jog feed or incremental feed is
0: Performed at feed per minute.
1: Performed at feed per rotation.

> **NOTE**
> Specify a feedrate in parameter No. 1423.

1423	Feedrate in manual continuous feed (jog feed) for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
(1) When bit 4 (JRV) of parameter No. 1402 is set to 0 (feed per minute), specify a jog feedrate (feed per minute) under an override of 100%.
(2) When bit 4 (JRV) of parameter No. 1402 is set to 1 (feed per revolution), specify a jog feedrate (feed per revolution) under an override of 100%.

> **NOTE**
> This parameter is clamped to the axis-by-axis manual rapid traverse rate (parameter No. 1424).

	#7	#6	#5	#4	#3	#2	#1	#0
1611							THA	

[Input type] Parameter input
[Data type] Bit path

#1 THA When a threading command is specified in AI contour control mode:
0: An alarm is issued.
1: AI contour control mode is temporarily canceled and the command is executed.

> **NOTE**
> 1 If look-ahead acceleration/deceleration before interpolation is enabled by the command following or followed by a threading command with this parameter set to 1, the tool is decelerated to a stop at the joint of the relevant block.
> 2 This parameter is automatic setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

	#7	#6	#5	#4	#3	#2	#1	#0
1612				MTA				

[Input type] Parameter input
[Data type] Bit path

#4 MTA When the maximum cutting feedrate is exceeded in threading, the alarm PS0530 is:
0: Issued.

1: Not issued.

	#7	#6	#5	#4	#3	#2	#1	#0
3405					G36			

[Input type] Parameter input
[Data type] Bit path

#3 G36 As a G code to be used with the automatic tool length measurement function (M series)/automatic tool offset function (T series) is:
0: G36 (T series only)/G37 is used.
1: G37.1/G37.2/G37.3 is used.

> **NOTE**
> If it is necessary to perform circular threading G36 (counterclockwise), set this parameter to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
3406								C01

[Input type] Parameter input
[Data type] Bit

#0 C01 When bit 6 (CLR) of parameter No. 3402 is 1, the RESET key on the MDI panel, the external reset signal, the reset and rewind signal, or emergency stop signal will,
0: Clear the G code with group number 01.
1: Not clear the G code with group number 01.

	#7	#6	#5	#4	#3	#2	#1	#0
3451								
								GQS

[Input type] Parameter input
[Data type] Bit path

#0 GQS When threading is specified, the threading start angle shift function (Q) is:
0: Disabled.
1: Enabled.

3708		TSO					SAT	SAR
		TSO						SAR

[Input type] Parameter input
[Data type] Bit path

#0 SAR The spindle speed arrival signal SAR<Gn029.4> is:
0: Not checked
1: Checked

#1 SAT Check of the spindle speed arrival signal at the start of executing the thread cutting block
0: The signal is checked only when bit 0 (SAR) of parameter No. 3708 is set to 1.
1: The signal is always checked irrespective of the setting of SAR.

> **NOTE**
> When thread cutting blocks are consecutive, the spindle speed arrival signal is not checked for the second and subsequent thread cutting blocks.

#6 TSO During a threading or tapping cycle, the spindle override is:
0: Disabled (tied to 100%).
1: Enabled.

> **NOTE**
> During rigid tapping, the override is tied to 100%, irrespective of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3713		MPC						

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#6 MPC When a spindle is selected with address P in a program during multi-spindle control (bit 3 (MPP) of parameter No. 3703 is set to 1), position coder feedback used for thread cutting, feed per revolution, and so forth is:
0: Not changed automatically according to the selected spindle.
1: Changed automatically according to the selected spindle.

> **NOTE**
> Setting this parameter produces the same effects as when position coder select signals PC2SLC<Gn028.7>, PC3SLC<Gn026.0>, and PC4SLC<Gn026.1>, inter-path spindle feedback signals SLPCA<Gn064.2>, SLPCB<Gn064.3>, SLPCC<Gn403.4>, and SLPCD<Gn403.5> are set.
> At this time, even when an attempt to set these signals is made by a PMC ladder, these signal operations are ignored.

	#7	#6	#5	#4	#3	#2	#1	#0
11602			NCP					

[Input type] Parameter input
[Data type] Bit path

#5 NCP If there is a non-threading block between two threading blocks, the second threading block:
0: Waits until the spindle one-rotation signal and the spindle speed arrival signal SAR <Gn029.4> are detected.
1: Does not wait until the spindle one-rotation signal and the spindle speed arrival signal SAR <Gn029.4> are detected unless a G code in non-threading group 01 is issued. (FS16*i* compatible specification)

Alarm and message

Number	Message	Description
PS0050	CHF/CNR NOT ALLOWED IN THRD BLK	Chamfering or corner R is commanded in the thread cutting block. Modify the program.
PS0429	ILLEGAL COMMAND IN G10.6	When retraction was started in the threading block, the retract command was performed in the longitudinal direction of the threading.
PS0509	TOOL OFFSET COMMAND IS NOT AVAILABLE	- Tool offset (for the lathe system) was specified in the thread cutting block. - Tool offset was specified (lathe system) in the scaling mode, coordinate system rotation mode, or programmable mirror image mode.
PS0530	EXCESS VELOCITY IN THREADING	Feedrate exceeds the maximum cutting feedrate.

Warning

⚠ **WARNING**

During threading, stopping feed without stopping the spindle is dangerous because the cutting depth will abruptly increase. Feed hold is, therefore, disabled during threading. If attempted during threading, feed stops in the same way as single block stop upon the completion of the first non-threading block after the termination of threading mode. Feed hold lamp signal SPL <Fn000.4>, however, turns to “1” immediately after feed hold signal *SP <Gn008.5> is set to “1”. The signal SPL turns to “0” when feed stops (the CNC enters the single block stop state).

Caution

⚠ **CAUTION**

1 Detector with one-rotation signal is required for threading, and it must be connected with spindle at a gear ratio of 1:1.
2 Feedrate override is ignored during threading, 100% being assumed.
3 During threading, spindle override is ignored, 100% being assumed.
4 When the first non-threading block is executed after threading mode has been finished, and the feed hold button is pressed again (or the feed hold button has been held down), the execution of the non-threading block is stopped immediately.
5 When threading is executed in the single block status, the tool stops after execution of the first block not specifying threading.
6 When the previous block was a threading block, cutting will start immediately without waiting for detection of the 1-turn signal even if the present block is a threading block.
7 When a dry run operation is performed the dry run rate becomes the longitudinal axis feedrate.
8 The threading retract function is supported only for the threading cycle.

Note

NOTE

Even in the machine lock state, threading detects a one-rotation signal from the position coder mounted to the spindle.
If threading is specified, and the threading is to be performed without waiting for a one-rotation signal from the position coder, it is necessary to enable dry run (bit 5 (TDR) of parameter No. 1401 = 0) with the threading command and set the dry run signal DRN <Gn046.7> to ”1”.
The threading command remains the same regardless of whether the threading cycle is a canned cycle or a multiple repetitive cycle.
Even if "without position coder feed per revolution" is used (bit 0 (NPC) of parameter No. 1402 = 1), set the dry run signal DRN to ”1” for threading in the machine lock state.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (For Lathe System) (B-64694EN-1)	Threading (G32)
OPERATOR’S MANUAL (For Machining Center System) (B-64694EN-2)	Threading (G33)
FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)	PARAMETERS RELATED TO DETECTORS

6.5.2 Threading Cycle Retract

T

When feed hold is performed during threading, the tool immediately retracts while performing chamfering and then returns to the start point. This function is valid for only canned cycle and multiple repetitive canned cycle as following table. This function is invalid for threading, variable lead threading, and circular threading.

	Machining center system	Lathe system			Threading Cycle Retract
		G code system			
		A	B	C	
Canned Cycle	-	G92	G78	G21	Valid
Multiple Repetitive Canned Cycle	-	G76	G76	G78	Valid
Threading	G33	G32	G33	G33	Invalid
Variable Lead Threading	G34	G34	G34	G34	Invalid
Circular Threading	G35,G36	G35,G36	G35,G36	G35,G36	Invalid
Circular thread cutting B	G02.1, G03.1	-	-	-	Invalid

The detail of this function is described below.

6.5.2.1 Threading cycle retract (canned cycle)

T

Overview

When feed hold is performed during threading in threading cycle (G92), the tool immediately retracts while performing chamfering and then returns to the start point in the order of the plane second axis (X-axis) followed by the plane first axis (Z-axis).

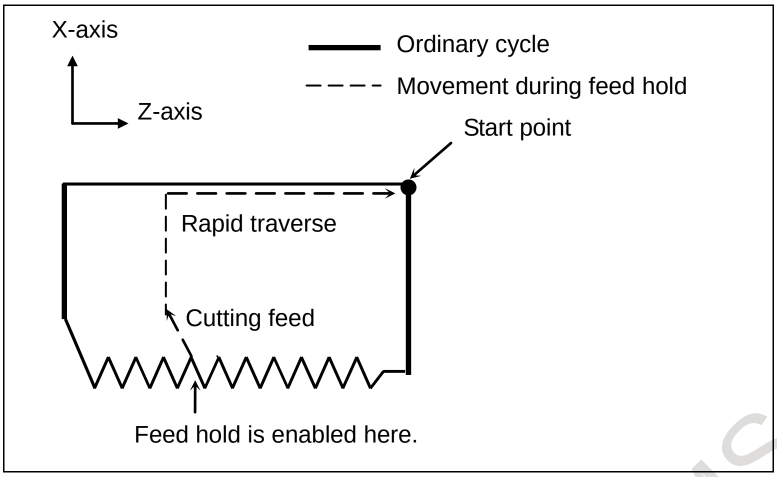


Fig. 6.5.2.1 (a)

The chamfering angle during retraction is the same as that at the end point.

- Thread chamfering

For the thread chamfering angle, 1° to 89° can be specified with parameter No. 5131. When the parameter value is 0, 45° is set.
The type of acceleration/deceleration after interpolation, time constant for acceleration/deceleration after interpolation, and FL feedrate used for thread chamfering are the same as those for threading.

- Retraction after chamfering

For retraction after chamfering, the feedrate, type of acceleration/deceleration after interpolation, and time constant are used as described in the Table 6.5.2.1 (a).

Table 6.5.2.1 (a)

Bit 0 (CFR) of parameter No.1611	Parameter No.1466	Description
0	Non-0	The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No.1626), FL feedrate (parameter No.1627), and feedrate for retraction (parameter No.1466).
0	0	The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No.1626), FL feedrate (parameter No.1627), and rapid traverse rate (parameter No.1420).
1		Before retraction a check is made to see that the specified feedrate has become 0 (delay in acceleration/deceleration is 0), and the type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant and the rapid traverse rate (parameter No. 1420).

For the feedrate for retraction after chamfering, the rapid traverse override can be disabled by setting 1 to bit 4 (ROC) of parameter No.1403.

> **NOTE**
> During retraction, the tool does not stop at cutting feedrate override of 0%, regardless of the setting of bit 4 (RF0) of parameter No.1401.

- Rapid traverse override

Rapid traverse override can be disabled during threading cycle retract by setting 1 to bit 7 (RTV) of parameter No.1403.

- Threading signal

Threading signal THRD<Fn002.3> turns to "1" in the block for returning to starting point of threading cycle by setting 1 to bit 1 (TRS) of parameter No.11223.

Signal

Threading signal THRD<Fn002.3>

[Function] This signal indicates that threading is in progress.

[Output cond.] This signal turns to "1" in the following cases:

- Threading mode in progress
- Threading cycle for turning

This signal turns to "0" in the following case.

- Neither threading mode nor thread cutting cycle are in progress.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1403	RTV			ROC				

[Input type] Parameter input
[Data type] Bit path

#4 ROC In the threading cycles G92 and G76 rapid traverse override for retraction after threading is finished is:
0: Effective
1: Not effective (Override of 100%)

#7 RTV Rapid traverse override while the tool is retracting in threading
0: Rapid traverse override is effective.
1: Rapid traverse override is not effective.

1466	Feedrate for retraction in threading cycle G92 or G76

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

When threading cycle G92 (T series) or G76 (T series) is specified, retraction is performed after threading. Set a feedrate for this retraction.

> ⚠ **WARNING**
> When the manual handle interruption is valid, set the same value as the parameter No.1430 to the parameter No.1466.

> **NOTE**
> When this parameter is set to 0 or bit 1 (CFR) of parameter No. 1611 is set to 1, the rapid traverse rate set in parameter No. 1420 is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1610			THLx					

[Input type] Parameter input
[Data type] Bit axis

#5 THLx Acceleration/deceleration in threading cycles G92 and G76
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied.
(Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)
As the time constant and FL rate, however, the settings of parameters Nos. 1626 and 1627 for threading cycles are used.

	#7	#6	#5	#4	#3	#2	#1	#0
1611								CFR

[Input type] Parameter input
[Data type] Bit path

#0 CFR For retraction after threading in the threading cycles G92 and G76:
0: The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No. 1626) and FL rate (parameter No. 1627).
1: The type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant.

NOTE
If this parameter is set to 1, a check is made before a retraction to see that the specified feedrate has become 0 (the delay in acceleration/deceleration has become 0). For retraction, the rapid traverse rate (parameter No. 1420) is used, regardless of the setting of parameter No. 1466. When this parameter is set to 0, parameter No. 1466 is used as the feedrate for retraction. As acceleration/deceleration used for retraction, only acceleration/deceleration after interpolation is used. Rapid traverse acceleration/deceleration before interpolation and optimum torque acceleration/deceleration are disabled.

1626	Acceleration/deceleration time constant in threading cycles for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
Set a time constant for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis.

1627	FL rate for acceleration/deceleration in threading cycles for each axis

[Input type] Parameter input

[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set an FL rate for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis. Set 0 at all times except in a special case.

5131	Cutting angle in thread cutting cycles G92 and G76

[Input type] Parameter input
[Data type] Byte path
[Unit of data] deg
[Valid data range] 1 to 89
This parameter sets a thread cutting angle in a thread cutting cycle (G92 (T series)/G76 (T series)).
When 0 is set, an angle of 45 degrees is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
11223							TRS	

[Input type] Parameter input
[Data type] Bit path

#1 TRS In threading cycle retraction, when a block that specifies return to the start point of the threading cycle is executed, threading signal THRD <Fn002.3> is:
0: Set to “0”.
1: Set to “1”.

Caution

> ⚠ **CAUTION**
> Feed hold cannot be performed during retracting.

Note

> **NOTE**
> The chamfering angle for retraction is determined by the setting of parameter No. 5131.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Threading cycle

6.5.2.2 Threading cycle retract (multiple repetitive canned cycle)

T

Overview

When feed hold is performed during threading in multiple repetitive threading cycle G76, threading is curtailed (chamfered) and then the tool returns to the start point in the threading cycle and stops.
At this time, if the cycle is started, it resumes from the threading cycle for which feed hold was applied.

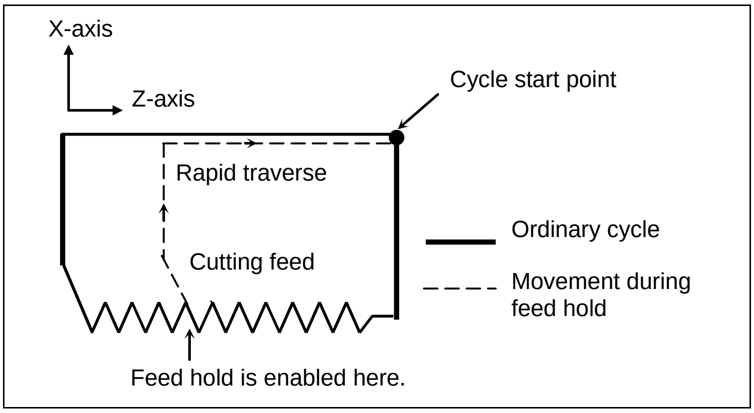


Fig. 6.5.2.2 (a)

The angle of chamfering during retracting is same as that at the end point.

- Thread chamfering

For the thread chamfering angle, 1° to 89° can be specified with parameter No. 5131. When the parameter value is 0, 45° is set.
The type of acceleration/deceleration after interpolation, time constant for acceleration/deceleration after interpolation, and FL feedrate used for thread chamfering are the same as those for threading.

- Retraction after chamfering

For retraction after chamfering, the feedrate, type of acceleration/deceleration after interpolation, and time constant are used as described in the Table 6.5.2.2(a).

Table 6.5.2.2 (a)

Bit 0 (CFR) of parameter No. 1611	Parameter No. 1466	Description
0	Non-0	The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No.1626), FL feedrate (parameter No.1627), and feedrate for retraction (parameter No.1466).
0	0	The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No.1626), FL feedrate (parameter No.1627), and rapid traverse rate (parameter No.1420).
1		Before retraction a check is made to see that the specified feedrate has become 0 (delay in acceleration/deceleration is 0), and the type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant and the rapid traverse rate (parameter No. 1420).

For the feedrate for retraction after chamfering, the rapid traverse override can be disabled by setting 1 to bit 4 (ROC) of parameter No.1403.

NOTE
During retraction, the tool does not stop at cutting feedrate override of 0%, regardless of the setting of bit 4 (RF0) of parameter No.1401.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1403	RTV			ROC				

[Input type] Parameter input
[Data type] Bit path

#4 ROC In the threading cycles G92 and G76, rapid traverse override for retraction after threading is finished is:
0: Effective
1: Not effective (Override of 100%)

#7 RTV Rapid traverse override while the tool is retracting in threading
0: Rapid traverse override is effective.
1: Rapid traverse override is not effective.

1466	Feedrate for retraction in threading cycle G92 or G76

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
When threading cycle G92 and G76 is specified, retraction is performed after threading. Set a feedrate for this retraction.

> ⚠ **WARNING**
> When the manual handle interruption is valid, set the same value as the parameter No.1430 to the parameter No.1466.

> **NOTE**
> When this parameter is set to 0 or bit 1 (CFR) of parameter No. 1611 is set to 1, the rapid traverse rate set in parameter No. 1420 is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1610			THLx					

[Input type] Parameter input
[Data type] Bit axis

#5 THLx Acceleration/deceleration in threading cycles G92 and G76
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied.
(Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)
As the time constant and FL rate, however, the settings of parameters Nos. 1626 and 1627 for threading cycles are used.

	#7	#6	#5	#4	#3	#2	#1	#0
1611								CFR

[Input type] Parameter input
[Data type] Bit path

#0 CFR For retraction after threading in the threading cycles G92 and G76:

0: The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No. 1626) and FL rate (parameter No. 1627).
1: The type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant.

NOTE
If this parameter is set to 1, a check is made before a retraction to see that the specified feedrate has become 0 (the delay in acceleration/deceleration has become 0). For retraction, the rapid traverse rate (parameter No. 1420) is used, regardless of the setting of parameter No. 1466. When this parameter is set to 0, parameter No. 1466 is used as the feedrate for retraction. As acceleration/deceleration used for retraction, only acceleration/deceleration after interpolation is used. Rapid traverse acceleration/deceleration before interpolation and optimum torque acceleration/deceleration are disabled.

1626	Acceleration/deceleration time constant in threading cycles for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
Set a time constant for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis.

1627	FL rate for acceleration/deceleration in threading cycles for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set an FL rate for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis. Set 0 at all times except in a special case.

5131	Cutting angle in thread cutting cycles G92 and G76

[Input type] Parameter input
[Data type] Byte path
[Unit of data] deg
[Valid data range] 1 to 89
This parameter sets a thread cutting angle in a thread cutting cycle (G92 and G76).
When 0 is set, an angle of 45 degrees is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
11223							TRS	

[Input type] Parameter input
[Data type] Bit path

#1 TRS In threading cycle retraction, when a block that specifies return to the start point of the threading cycle is executed, threading signal THRD <Fn002.3> is:
0: Set to “0”.
1: Set to “1”.

Caution

> ⚠ **CAUTION**
> Feed hold cannot be performed during retracting.

Note

> **NOTE**
> The chamfering angle for retraction is determined by the setting of parameter No. 5131.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Threading cycle

6.5.3 Variable Lead Threading

T

Overview

Specifying an increment or a decrement value for a lead per screw revolution in G34 command enables variable-lead thread cutting to be performed.

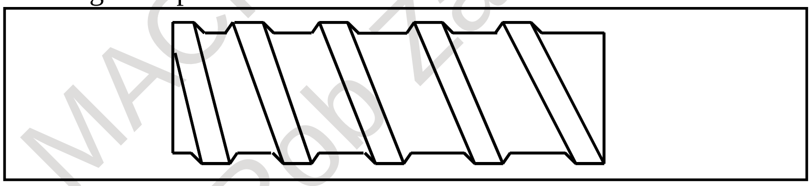


Fig. 6.5.3 (a)

> **NOTE**
> The "thread cutting cycle retract" is not effective for G34.

Alarm and message

Number	Message	Description
PS0313	ILLEGAL LEAD COMMAND	The variable-lead threading increment specified in address K exceeds the specified maximum value in variable-lead threading. Or, a negative lead value was specified.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Variable lead threading

6.5.4 Continuous Threading

Overview

Threading blocks can be programmed successively to eliminate a discontinuity due to a discontinuous movement in machining by adjacent blocks.

Since the system is controlled in such a manner that the synchronism with the spindle does not deviate in the joint between blocks wherever possible, it is possible to performed special threading operation in which the lead and shape change midway.

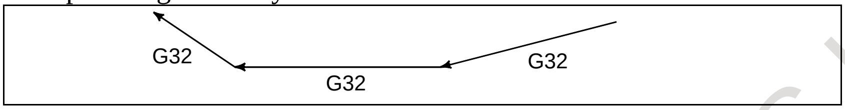


Fig. 6.5.4 (a)

Even when the same section is repeated for thread cutting while changing the depth of cut, this system allows a correct machining without impairing the threads.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Continuous treading

6.5.5 Multiple Threading

Overview

Using the Q address to specify an angle between the one-spindle-rotation signal and the start of threading shifts the threading start angle, making it possible to produce multiple-thread screws with ease.

M

Set the bit 0 (GQS) of parameter No. 3451 to 1 for using this function.

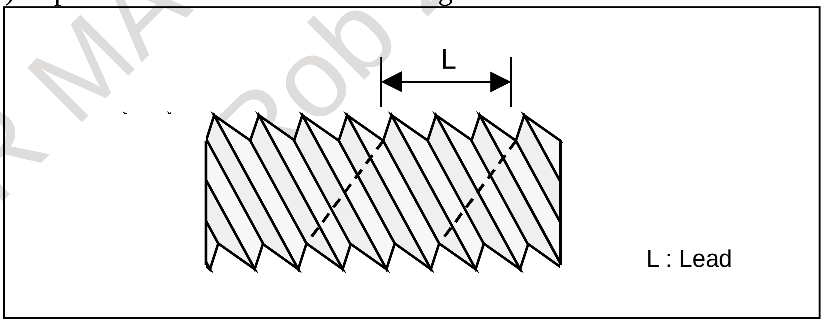


Fig. 6.5.5 (a)

Limitation

- Threading start angle

The threading start angle is not a continuous state (modal) value. It must be specified each time it is used. If a value is not specified, 0 is assumed.

- Threading start angle increment

The threading start angle (Q) increment is 0.001 degrees. Note that no decimal point can be specified.
Example:

For a shift angle of 180 degrees, specify Q180000.
Q180.000 cannot be specified, because it contains a decimal point.

Note: Q1 is the command of 0.001 degree for the angle for shifting the threading start angle, regardless of the setting of the followings.

- Increment system IS-A/B/C (Parameter No.1013#3-#0)
- Pocket calculator type decimal point programming (Bit 0(DPI) of parameter No.3401)
- The least input increment is 10 times greater than the least command increment (Bit 7(IPR) of parameter No.1004)

- Specifiable threading start angle range

The threading start angle (Q) of between 0 and 360000 (in 0.001-degree units) can be specified. If a value greater than 360000 (360 degrees) is specified, it is rounded down to 360000 (360 degrees).
If a minus value is specified, it works as a plus value.
Example:

If Q-90000 (-90 degrees) is specified, it works as Q90000 (90 degrees).

T

- Multiple threading cycle (G76 (G code system A/B)) (G78 (G code system C))

The address Q of the G76/G78 multiple threading cycle command is used for the minimum cutting depth or the depth of cut in 1st cut. For this reason, the angle for shifting the threading start angle can not be commanded.
However, if the FS15 tape format is used, in G76/G78 multiple threading cycle, the address Q is possible to specify the angle for shifting the threading start angle.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3451								
								GQS

[Input type] Parameter input
[Data type] Bit path

#0 GQS When threading is specified, the threading start angle shift function (Q) is:
0: Disabled.
1: Enabled.

6.5.6 Circular Threading

T

Overview

Using the G35 and G36 commands, a circular thread, having the specified lead in the direction of the major axis, can be machined.

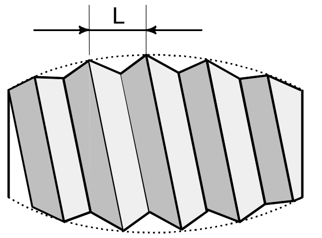


Fig. 6.5.6 (a)

> **NOTE**
> The "thread cutting cycle retract" is not effective for G35, G36.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3405					G36			

[Input type] Parameter input
[Data type] Bit path

#3 G36 As a G code to be used with the automatic tool length measurement function (M series)/automatic tool offset function (T series) is:
0: G36 (T series only)/G37 is used.
1: G37.1/G37.2/G37.3 is used.

> **NOTE**
> If it is necessary to perform circular threading G36 (counterclockwise), set this parameter to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
3451								GQS

[Input type] Parameter input
[Data type] Bit path

#0 GQS When threading is specified, the threading start angle shift function (Q) is:
0: Disabled.
1: Enabled.

Alarm and message

Number	Message	Description
PS5058	G35/G36 FORMAT ERROR	A command for switching the major axis has been specified for circular threading. Alternatively, a command for setting the length of the major axis to 0 has been specified for circular threading.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Circular threading

6.5.7 Arbitrary Speed Threading

6.5.7.1 Arbitrary speed threading

Overview

In conventional threading, spindle speed could not be changed because tool movement could not synchronize with spindle rotation, and machining accuracy of a screw is decreased. This function makes it possible to change spindle speed during threading. This function can be used with threading, threading cycle, and multiple threading cycle. Changing spindle speed can prevent the vibration that occurs during threading at the specific spindle speed for a large size machine. Besides, for repetitive machining, the same thread shape can be machined even if spindle speed is changed between rough machining and finishing machining. Cs contour control is required for this function.

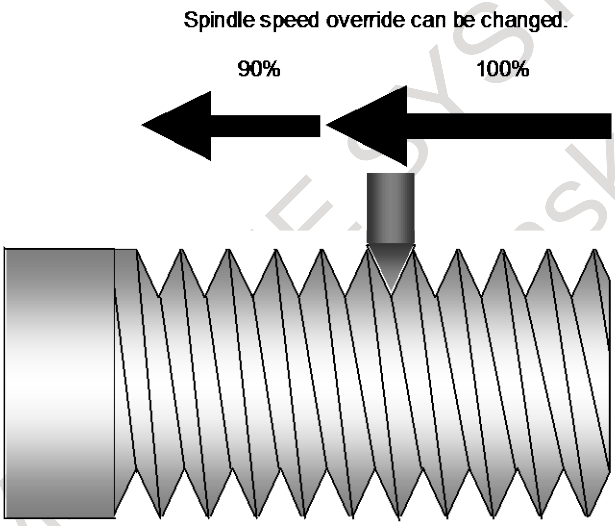


Fig.6.5.7.1 (a) Arbitrary speed threading

Furthermore, even if a workpiece is removed from a chuck before threading is not finished, re-machining of this workpiece can be achieved. Refer to the section "6.5.7.2 Re-machining thread" for details.

Format

Mxx ;	**Start of arbitrary speed threading**
G32IP_F_;	**Threading command**
Myy ;	**Cancel of arbitrary speed threading**

xx : M code set in parameter No.11487
yy : M code set in parameter No.11488

Explanation

The system enters arbitrary speed threading mode according to the sequence of arbitrary speed threading mode change by specifying the M code set in parameter No.11487 (start of arbitrary speed threading). Then, threading (threading, threading cycle, or multiple threading cycle) is specified. Arbitrary speed threading mode is canceled by specifying the M code set in parameter No.11488 (cancel of arbitrary speed threading). Arbitrary speed threading is enabled by setting bit 0 (AST) of parameter No.11485 to 1.
Example of machining program

```
N1G00 X10.Z50.;
N2S_;                         : specifying the spindle speed
N3Mxx;                        : start of arbitrary speed threading
N4G92X151.0Z-200.F10.;        : start of threading cycle
N5X150.0;
N6X149.0;
N7X148.0;
N8G00;                        : cancel of threading cycle
N9Myy;                        : cancel of arbitrary speed threading
```

The same thread shape can be machined in threading cycle specified N4 to N7 blocks, even if the spindle speed is changed.

NOTE
1 M code for arbitrary speed threading must be specified alone in a block.
2 M code for arbitrary speed threading prevents buffering.
3 Start command of arbitrary speed threading must be specified in canned cycle cancel mode for drilling. .
4 M code of arbitrary speed threading cancel must be specified after canceling threading mode. If the M code is specified in threading mode, the alarm PS0529, “THREADING COMMAND IMPOSSIBLE” is issued.
5 During arbitrary speed threading mode, feedback pulse of a position coder must be the one from the spindle which is commanded Mxx (xx is set in parameter No.11487).
6 If threading, threading cycle, or multiple threading cycle is specified when arbitrary speed threading mode is canceled, usual threading is executed.

- Command in arbitrary speed threading mode

Arbitrary speed threading can be used with the following threading commands.

Table 6.5.7.1 (a) Threading commands used with arbitrary speed threading

	Machining center system	Lathe system		
		G code system		
		A	B	C
Threading	G33	G32	G33	G33
Threading cycle	–	G92	G78	G21
Multiple threading cycle	–	G76	G76	G78

NOTE

1. When continuous threading is executed, the lead along the feed axis (the first axis on the plane) must not be changed between threading blocks.
2. If the next block of threading command (G32: lathe system in G code system A, G33: machining center system) does not include threading, the feedrate is decelerated at the end point of threading command.
3. If a taper angle is lower than that of the previous block in continuous threading, or if a chamfering angle is lower than the taper angle in threading cycle, the alarm PS0529 is issued. Machining program must be created as the direction of chamfering during arbitrary speed threading is in the hatching area of Fig. 6.5.7.1 (b).

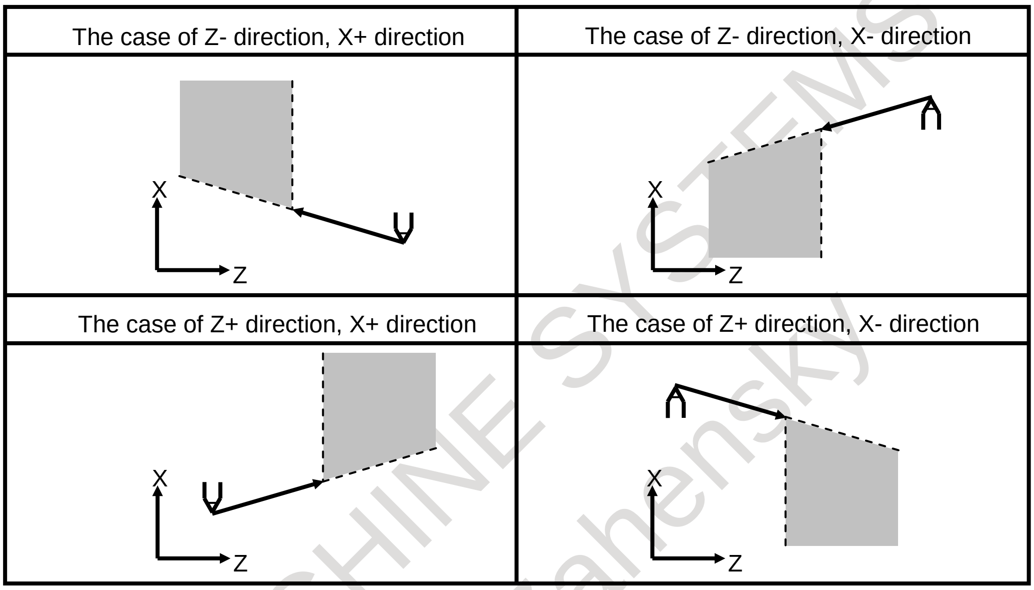


Fig. 6.5.7.1 (b) Chamfering during arbitrary speed threading

- Change of arbitrary speed threading mode

Cs contour control high speed switching signals are used for arbitrary speed threading. The spindle control mode is switched from spindle rotation control to Cs contour control without stopping spindle rotation when arbitrary speed threading mode is selected. To cancel arbitrary speed threading mode, Cs contour control change signal CON <Gn027.7> or Cs contour control change signals in each axis CONS <Gn274.0 to 3> is set to "0" in M code of canceling arbitrary speed threading, then, spindle control mode is switched from Cs contour control to spindle rotation control without stopping spindle rotation. The following parameter setting is required to change arbitrary speed threading mode.

Table 6.5.7.1 (b) Parameters to set for change of arbitrary speed threading mode

Parameter number	Description
ROT(No.1006#0)	Type of rotary axis(Please set to 1)
ROS(No.1006#1)	Type of rotary axis(Please set to 0)
ROAx(No.1008#0)	Whether to use the roll-over function of a rotary axis(Please set to 1)
RRLx(No.1008#2)	Whether Relative coordinates to be rounded by the amount of shift per rotation(Please set to 1)
CSDRCT(No.4002#4)	Whether to use the rotation direction signal (SFR/SRV) on Cs contouring control(Please set to 1)
No.4504	Cs contour control high speed switching : Reference position establishment speed
No.4505	Cs contour control high speed switching : Reference position establishment acceleration

Parameter number	Description
No.4506	Cs contour control high speed switching : Bell shaped acceleration/deceleration time constant for the reference position establishment
No.4507	Cs contour control high speed switching : Reference position establishment completion level
AST(No.11485#0)	Whether to use the function of arbitrary speed threading
No.11487	M code to start arbitrary speed threading mode
No.11488	M code to cancel arbitrary speed threading mode
No.11489	Acceleration in arbitrary speed threading

Rotation direction in Cs contour control may be different from that in spindle rotation control according to machine configuration or parameter setting. In this case, SV reverse signals SVRVS1 to SVRVS8 <Gn523> are set to “1” before changing to the arbitrary speed threading mode. Also when the arbitrary speed threading mode is canceled, SV reverse signal is set to “0”.

Example of reverse rotation direction in Cs contour control

-The case of the following machine configuration:

When the motor rotation direction is plus, the spindle rotation direction is minus owing to the gear between the motor and the spindle.

Forward rotation/Reverse rotation command signal SFR/SRV specifies the motor rotation direction in spindle rotation control, on the other hand these signals specify the spindle rotation direction in Cs contour control. Therefore, the relationship between command, motor rotation direction, and spindle rotation direction is as follows.

Spindle rotation control:
Command (+) – Motor rotation direction (+) – Spindle rotation direction (-)

Cs contour control:
Command (+) – Motor rotation direction (-) – Spindle rotation direction (+)

NOTE

1 When bit 1 (ART) of parameter No.11486 is set to 0, if reset operation is executed in arbitrary speed threading mode, Cs contour control change signal CON <Gn027.7> or Cs contour control change signals in each axis CONS <Gn274.0 to 3> must be set to “0” to cancel arbitrary speed threading mode. If an automatic operation is restarted without performing this operation, alarm PS0529 “THREADING COMMAND IMPOSSIBLE” is issued. When bit 1 (ART) of parameter No.11486 is set to 1, if reset operation is executed, arbitrary speed threading mode is maintained. To cancel arbitrary speed threading mode, the arbitrary speed threading mode cancellation M code (parameter No.11488) is specified, and Cs contour control change signal CON <Gn027.7> or Cs contour control change signals in each axis CONS <Gn274.0 to 3> must be set to “0”.

2 Cs contour control change signal CON must be set to “1” after the speed arrival signal SARA to SARD <Fn045.3, Fn049.3, Fn168.3, Fn266.3> become “1” when arbitrary speed threading mode is changed.

3 SV reverse signal setting must be decided after checking the rotation direction in spindle rotation control and that in Cs contour control by changing arbitrary speed threading mode with a speed of 0[min-1] (S0) once.

4 Reference position return of Cs contour control axis must be performed once after power-on to detect one-rotation signal of a spindle.

- Sequence of arbitrary speed threading mode change

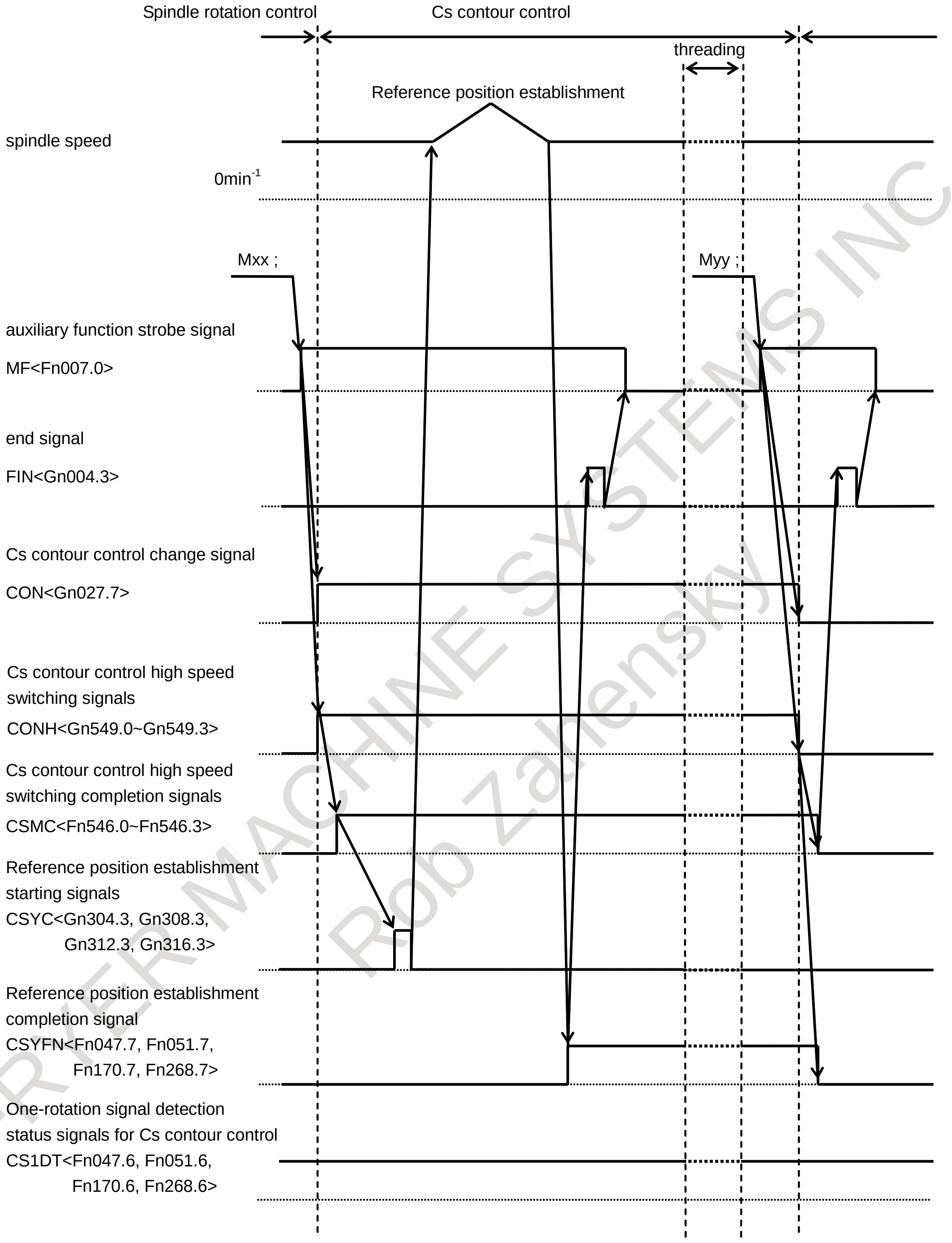


Fig.6.5.7.1 (c) sequence of arbitrary speed threading mode change

- Spindle speed override

Spindle speed override signals SOV0 to SOV7 <Gn030> are available. If the override for individual spindles is enabled, SOV20 to SOV27 <Gn376> (second spindle), SOV30 to SOV37 <Gn377> (third spindle), SOV40 to SOV47 <Gn378> (forth spindle) are available, too.
However, changing the spindle speed override is disabled during the following period.

- After the Cs contour control change signal CON is set to “1” until the reference position establishment completion signal CYSFN becomes “1”.
- After threading starts until the feedrate of feed axis (the first axis on the plane) arrives at the commanded speed.

NOTE

1 If spindle speed override in each path is used, not only the spindle speed of the spindle in arbitrary speed threading mode but also the spindle speed of spindles in the same path is not changed according to spindle speed override signals.

2 As the spindle speed gets higher, the incorrect thread length is longer. Machining program must be created considering the incorrect thread length which is changed according to the range of spindle speed override in arbitrary speed threading mode.

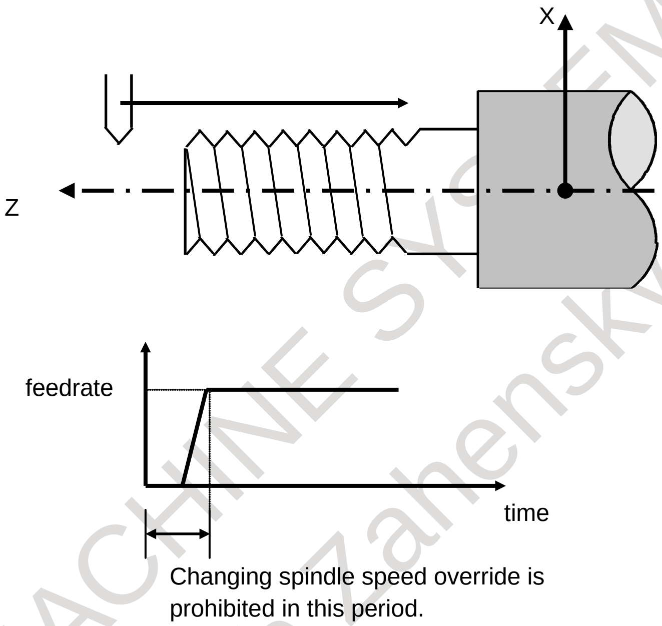


Fig.6.5.7.1 (d) Period in which changing spindle speed override is prohibited

- Acceleration/Deceleration of spindle in arbitrary speed threading

It is possible to switch the acceleration of a spindle according to spindle speed when the spindle speed override is changed in arbitrary speed threading. There are two switching points, which is set in parameter No.11020 and No.11021. Besides, acceleration of each section is set in parameter No.11030, No.11031, and No.11032.

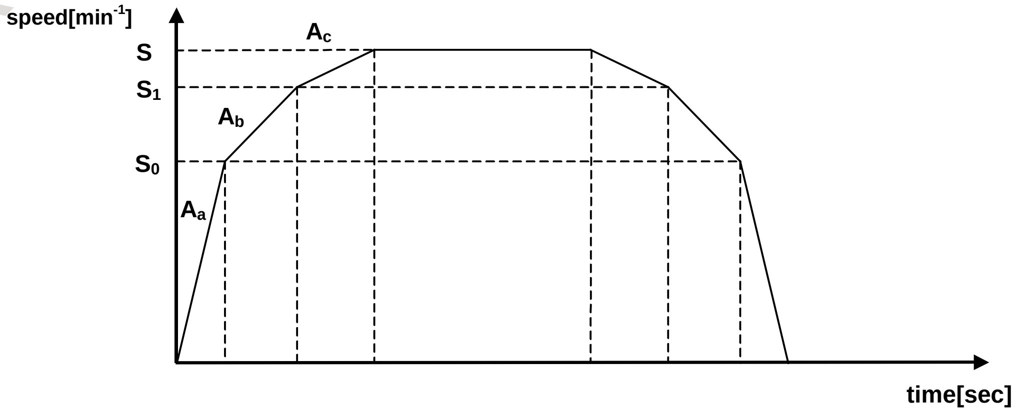


Fig.6.5.7.1 (e) Acceleration/deceleration of spindle in arbitrary speed threading

S_0: value set in parameter No.11020 (Acceleration is switched at S_0[min^{-1}].)

S_1: value set in parameter No.11021 (Acceleration is switched at S_1[min^{-1}].)
S : specified rotation speed[min^{-1}]
A_a : value set in parameter No.11030 (Acceleration [min^{-1}/s] applied from 0[min^{-1}] to S_0[min^{-1}] (section 1))
A_b : value set in parameter No.11031 (Acceleration [min^{-1}/s] applied from S_0[min^{-1}] to S_1[min^{-1}] (section 2))
A_c : value set in parameter No.11032 (Acceleration [min^{-1}/s] applied from S_1[min^{-1}] to maximum speed (section 3))

In addition, the deceleration can be set in other parameter by setting bit 6 (DCS) of parameter No.11001 to 1.

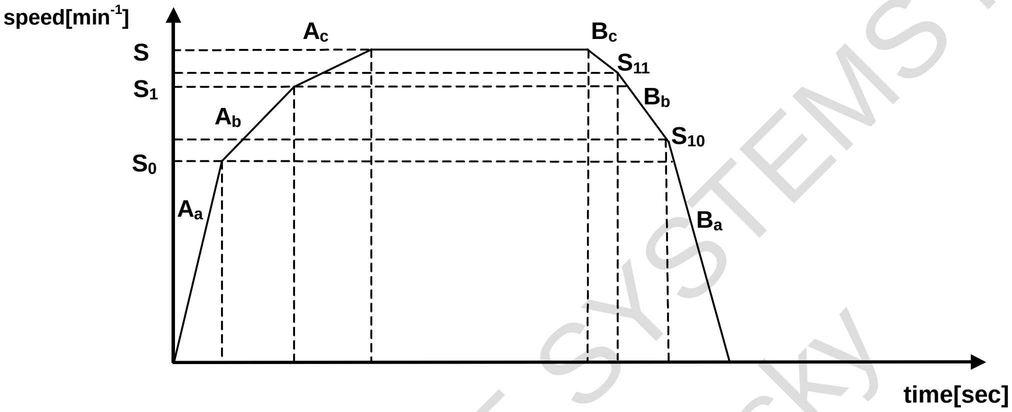


Fig.6.5.7.1 (f) Acceleration/deceleration of spindle in arbitrary speed threading (exclusive parameter for deceleration)

S_{10} : value set in parameter No.25700 (Deceleration is switched at S_{10}[min^{-1}].)
S_{11} : value set in parameter No.25701 (Deceleration is switched at S_{11}[min^{-1}].)
B_a : value set in parameter No.25710 (Deceleration [min^{-1}/s] applied from 0[min^{-1}] to S_{10} [min^{-1}] (section 1))
B_b : value set in parameter No.25711 (Deceleration [min^{-1}/s] applied from S_{10} [min^{-1}] to S_{11} [min^{-1}] (section 2))
B_c : value set in parameter No.25712 (Deceleration [min^{-1}/s] applied from S_{11} [min^{-1}] to maximum speed (section 3))

Determine the value of each parameter considering the torque characteristic of the motor and so on.

- Spindle speed at the start of threading

Spindle speed at the start of threading must be arrived at the specified speed in arbitrary speed threading. Therefore, threading can be started in the mode if the difference between specified speed and feedback pulse from a position coder is smaller than the level set in parameter No.11490.

> **NOTE**
> Speed arrival signals SARA to SARD <Fn045.3, Fn049.3, Fn168.3, Fn266.3> are always “1” while Cs contour control high speed switching completion signal is “1”.

- Acceleration/Deceleration of feed axis and infeed axis in threading

In threading, the axis whose travel distance is the longest is called a major axis. Thread is machined so that the specified lead would be applied in the direction of the major axis. In arbitrary speed threading, exclusive linear acceleration/deceleration is executed. Acceleration for the major axis is set in parameter No.11489. As for infeed axis (the second axis on the plane) performing taper threading and chamfering, acceleration is calculated from that of the major axis automatically. In continuous threading, acceleration of the major axis is applied from the first threading block to the last threading block. If the spindle speed override is changed, the spindle acceleration/deceleration is executed according to the switching point set in parameter No.11020, No.11021, and acceleration set in parameter No.11030 to No.11032, then, the major axis acceleration/deceleration is executed according to the spindle acceleration/deceleration.

Example: continuous threading
In block 1, acceleration for Z axis (major axis) is applied whose value is set in parameter No.11489, acceleration for X axis is calculated automatically. In block 2, the major axis is changed from Z axis to X axis because the chamfering angle is 60 degrees. But acceleration for Z axis is not changed (same as block 1), and acceleration for X axis is calculated automatically.

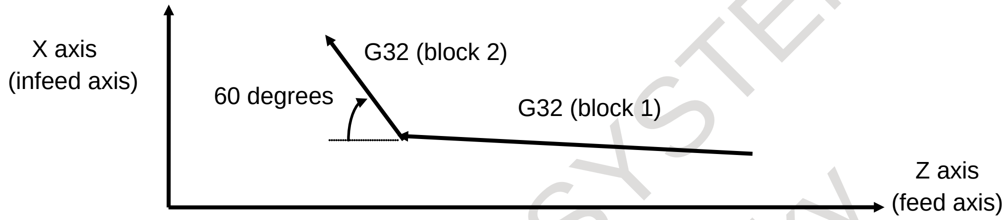


Fig. 6.5.7.1 (g) Acceleration/Deceleration of feed axis and infeed axis in threading

NOTE

1 Acceleration of a major axis must be higher than that of a spindle in order to machine without breaking the thread even if the spindle speed is changed. Acceleration of a major axis (parameter No.11489) must be set so that the acceleration would satisfy the following inequality.

$A_{ax}>(A_{sp}/60)*F$

Where,

A_{ax} : acceleration of major axis set in parameter No.11489

A_{sp} : maximum acceleration of spindle set in parameters No.11030 to No.11032, No.25710 to No.25712

F : maximum lead specified in a machining program

If the above condition is not satisfied, alarm PS0531 “THREADING PARAMETER ERROR” is issued.

2 In continuous threading, if the feedrate of infeed axis (the second axis on the plane) cannot be decelerated to 0 with automatically calculated acceleration, alarm PS0529 is issued (For example when the major axis of block 1 and block 2 is the second axis on the plane, and travel distance of block 2 is short.).

Chamfering value and angle in thread cutting cycles

In usual threading, parameter No.5130 for chamfering value in thread cutting cycles and parameter No.5131 for chamfering angle in thread cutting cycles are used. In arbitrary speed threading, the parameter is selected to use in chamfering by chamfering for arbitrary speed threading signal ASTC<Gn549.6>.

- Chamfering value and angle in usual thread cutting cycles (Parameters No.5130, No.5131)
- Chamfering value and angle in thread cutting cycles for arbitrary speed threading (Parameters No.11497, No.11498).

NOTE
Use M code without buffering when ASTC <Gn549.6> is changed during arbitrary speed threading mode.

- Retraction after chamfering for threading cycle and multiple threading cycle

Feedrate, type of acceleration/deceleration, and time constant of retraction after chamfering (3. retraction of Fig. 6.5.7.1 (h) threading cycle) are as usual, as shown in Table 6.5.7.1 (c).

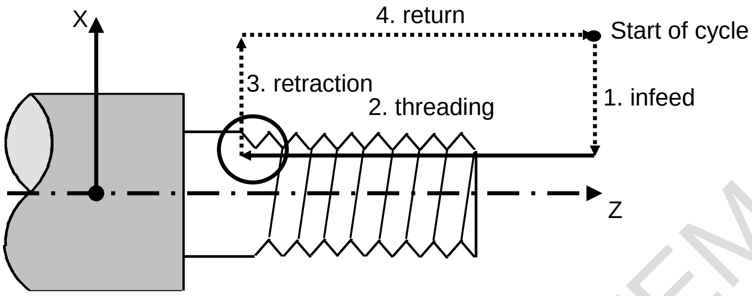


Fig. 6.5.7.1 (h) threading cycle

Table 6.5.7.1 (c) setting of acceleration/deceleration for retraction after chamfering

Parameter CFR (No.1611#0)	Parameter (No.1466)	Description
0	Other than 0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No.1626), FL feedrate (parameter No.1627), and retraction feedrate specified in parameter No.1466.
0	0	Uses the type of acceleration/deceleration after interpolation for threading, time constant for threading (parameter No.1626), FL feedrate (parameter No.1627), and rapid traverse rate specified in parameter No.1420.
1		Before retraction, a check is made to see that the specified feedrate has become 0 (delay in acceleration/deceleration is 0), and the type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant and the rapid traverse rate (parameter No.1420).

When the type of acceleration/deceleration after interpolation for threading is used as the acceleration / deceleration for retraction after chamfering (bit 0 (CFRx) of parameter No.1611 is 1), acceleration / deceleration after interpolation for threading is acceleration/deceleration of exponential interpolation type. By setting bit 5 (THLx) of parameter No.1610, the same acceleration/deceleration for cutting feed can be selected (Followed by the setting of bits 1 (CTBx) and 0 (CTLx) of parameter No.1610). However, as a time constant and FL feedrate, the settings of parameter No.1626 and No.1627 for the threading cycle are used.

NOTE
Acceleration/deceleration after interpolation is invalid during arbitrary speed threading (2.threading in Fig. 6.5.7.1 (h) threading cycle), and acceleration/deceleration is controlled according to the exclusive acceleration. Retraction is started after finishing the deceleration of feed axis and infeed axis.

- Maximum cutting feedrate

If the cutting feedrate of feed axis or infeed axis exceeds the maximum cutting feedrate (parameter No.1430), cutting feedrate is clamped to the limit set in parameter No.1430, and the alarm PS0530 “EXCESS VELOCITY IN THREADING” is issued. Threading is continued even if the alarm is issued.

Automatic operation is stopped after retracting in the case of threading cycle and multiple threading cycle. Automatic operation is stopped at the end of a block in the case of threading.

- Acceleration/Deceleration after interpolation

Acceleration/Deceleration after interpolation is ineffective in threading and chamfering.

- Dry run

Dry run is available according to the setting of bit 5 (TDR) of parameter No.1401.

- Position gain

Position gain of spindle (Cs contour control axis), position gain of feed axis, and position gain of infeed axis must be set the same value.

- feed axis, infeed axis
  parameter No.1825
- spindle (Cs contour control axis)
  parameter No.4069 (HIGH gear)
  parameter No.4070 (MEDIUM HIGH gear)
  parameter No.4071 (MEDIUM LOW gear)
  parameter No.4072 (LOW gear)

- Gear change system

T type gear selection method must be selected for the path which is applied arbitrary speed threading.

- Path spindle control

Start and cancel command of arbitrary speed threading must be specified in the same path. The path specified M code of starting arbitrary speed threading and the path to which a spindle for arbitrary speed threading must be set bit 0 (AST) of parameter No.11485 to 1. When the setting of bit 0 (AST) of parameter No.11485 is different because of the path specified M code of starting arbitrary speed threading and the path to which a spindle for arbitrary speed threading, it operates as follows.

- When the path specified M code is valid and the path to which a spindle is invalid:
  An alarm PS0531 "THREADING PARAMETER ERROR" occurs. M code is not output.
- When the path specified M code is invalid and the path to which a spindle is valid:
  It is not processed as M code for arbitrary speed threading. M code is output.

It is not able to specify the same spindle as the axis performing arbitrary speed threading among multiple paths. When the spindle is already used in arbitrary speed threading mode at one path, if the command of starting arbitrary speed threading is specified to the same spindle at other path, the alarm PS0529 "THREADING COMMAND IMPOSSIBLE" is issued.

Set parameters for arbitrary speed threading as follows.
Path A: The path specified M code of starting arbitrary speed threading
Path B: The path to which a spindle for arbitrary speed threading

Necessary parameters both path A and path B to set	
Parameter No.	**Explanation**
No.11485#0	Arbitrary speed threading is valid

Necessary parameter for path A to set	
Parameter No.	**Explanation**
No.11486#0	Command for shifting the threading start angle by address Q in re-machining thread is invalid (0) / valid (1)
No.11486#1	Arbitrary speed threading mode is canceled (0) / not canceled (1) by reset
No.11486#2	In arbitrary speed threading mode, M code to start arbitrary speed threading mode is invalid (0) / valid (1) to command
No.11489	Acceleration of major axis in arbitrary speed threading (set to major axis)

Necessary parameter for path A to set	
Parameter No.	**Explanation**
No.11490	Spindle speed arrival level in arbitrary speed threading
No.11492	Adjusting parameter 1 for arbitrary speed threading (position error of servo)
No.11493	Adjusting parameter 2 for arbitrary speed threading (position error of spindle)

Necessary parameter for path B to set	
Parameter No.	**Explanation**
No.11487	M code to start arbitrary speed threading mode
No.11488	M code to cancel arbitrary speed threading mode
No.11030	Individual acceleration/deceleration
No.11031	Individual acceleration/deceleration
No.11032	Individual acceleration/deceleration
No.4002#4	In Cs contour control, the rotation direction signal (SFR/SRV) function is used / not used. (Please set to 1)
No.4069	Position gain in Cs contour control (HIGH gear)
No.4070	Position gain in Cs contour control (MEDIUM HIGH gear)
No.4071	Position gain in Cs contour control (MEDIUM LOW gear)
No.4072	Position gain in Cs contour control (LOW gear)
No.4504	Cs contour control high speed switching: Reference position establishment speed
No.4505	Cs contour control high speed switching: Reference position establishment acceleration
No.4506	Cs contour control high speed switching: Bell-shaped acceleration/deceleration time constant for the reference position establishment
No.4507	Cs contour control high speed switching: Reference position establishment completion level

Note

- Emergency stop

A spindle (Cs contour control axis) is stopped by emergency stop *ESP <X008.4> (for the first machine group), <X008.0> (for the second machine group), <X008.1> (for the third machine group), and <Gn008.4> during arbitrary speed threading.

- Spindle stop

A spindle can be stopped by the following operations during arbitrary speed threading.

- Spindle stop signal *SSTP<Gn029.6>, individual spindle stop signal *SSTP1 to *SSTP4<Gn027.3 to Gn027.5, Gn026.6> (using multi-spindle control)
- Spindle speed override signal, individual spindle speed signal
- S0 command

Arbitrary speed threading must be canceled when a spindle is stopped by an operation except above during arbitrary speed threading. Besides, a spindle is stopped by servo alarm or servo-off differently from usual spindle control.

- Machine lock

All-axis machine lock signal MLK<Gn044.1> and each-axis machine lock signals MLK1 to MLK8<Gn0108> are ineffective for a spindle during arbitrary speed threading. These signals are effective before threading. But these signals are ineffective during threading including chamfering.

- Interlock

All axes interlock signal *IT<Gn008.0> and interlock signals for each axis *IT1 to *IT8<Gn130> are ineffective for a spindle during arbitrary speed threading.

- Stored stroke check

Stored stroke check is ineffective for a spindle during arbitrary speed threading.

- Arbitrary speed threading in Cs contour control

If the M code for start of arbitrary speed threading is specified in Cs contour control mode, the alarm (PS0529) ” THREADING COMMAND IMPOSSIBLE” is issued. Changing to arbitrary speed threading mode is permitted in only spindle rotation control mode.

- Simple spindle electronic gear box

The slave axis synchronizes by following to the master axis, therefore, the slave axis is required a higher acceleration ability than that of the master axis. If the acceleration ability of the master axis is higher than that of the slave axis, the acceleration/deceleration of the master axis should be set in consideration of the acceleration ability of the slave axis. Moreover, the spindle which is commanded in arbitrary speed threading mode should be a master axis.

- Spindle speed

When bit 3 (IRC) of parameter No.1408 is set to 0, the malfunction prevent function may work at a rotation speed of about 2778 min^{-1}. When the motor rotates at a rotation speed of 2778 min^{-1} or more, set bit 3 (IRC) of parameter No.1408 to 1. If bit 3 (IRC) of parameter No.1408 is set to 1, a rotation speed can be specified up to about 27778 min^{-1}.

Limitation

- Reference position return

As a rule, in order to use this function, reference position return must be performed in Cs contour control once after power-on to detect the one-rotation signal of the spindle. If arbitrary speed threading mode change is executed without performing reference position return, the spindle decelerates to the reference position return speed (parameter No.4074) in order to detect the one-rotation signal. After detecting the one-rotation signal, one-rotation signal detection status signal for Cs contour control CS1DTx (<Fn047.6> (1st spindle), <Fn051.6> (2nd spindle), <Fn170.6> (3rd spindle), <Fn268.6> (4th spindle)) become “1”.

- Unavailable functions for arbitrary speed threading

- Pitch error compensation
  Pitch error compensation is not available for spindle in arbitrary speed threading.
- Synchronous control and composite control
  Synchronous control and composite control are not available for the first axis on the plane, the second axis on the plane, and the spindle during arbitrary speed threading. Alarm PS0529 “THREADING COMMAND IMPOSSIBLE” is issued.
- Superimposed control
  Superimposed control is not available for the first axis on the plane, the second axis on the plane, and the spindle during arbitrary speed threading. Alarm PS0529 “THREADING COMMAND IMPOSSIBLE” is issued.
- Spindle synchronous control
- Spindle command synchronous control
  Both spindle synchronous control and spindle command synchronous control are not available for the spindle during arbitrary speed threading. Alarm PS0529 “THREADING COMMAND IMPOSSIBLE” is issued.
- Manual reference position return
  If manual reference position return for spindle is performed in arbitrary speed threading mode, alarm PS0529 “THREADING COMMAND IMPOSSIBLE” is issued.
- Arbitrary reference position setting function
- Cs contour control axis coordinate establishment
  Both arbitrary reference position setting function and Cs contour control axis coordinate establishment are not available for a spindle during arbitrary speed threading.
- Spindle control with a servo motor
  Arbitrary speed threading is not available for a spindle which is controlled with a servo motor.
- Spindle analog output

Arbitrary speed threading is not available for analog spindle.

- Usual threading
  Usual threading is not available during arbitrary speed threading mode. Arbitrary speed threading must be canceled to enable usual threading.
- Manual handle retrace
- Spindle output control by the PMC
- Balance cutting
- Mirror image for double turret
  Arbitrary speed threading is not available during double turret mirror image mode.
- Flexible path axis assignment
  Arbitrary speed threading is not available for a spindle which is assigned/exchanged by flexible path axis assignment.
- Constant surface speed control
  Arbitrary speed threading is not available in constant surface speed control.

- Spindle software functions

The following functions cannot be used with arbitrary speed threading.

- Spindle electronic gear box (spindle EGB)
- Dual position feedback function

6.5.7.2 Re-machining thread

Overview

This function makes it possible to re-machine the same thread even if the workpiece which is not finished threading is removed from a chuck once. In order to re-machine the thread, groove of thread is measured after mounting the workpiece on a chuck. Then, the thread is re-machined by the same machining program. This function is included in “Arbitrary speed threading”.

Explanation

In order to re-machine the thread, the groove of thread is measured. Then, re-machining thread signal RMTC<Gn549.5> is set to “1”, and re-machining thread is executed in arbitrary speed threading mode. This function is enabled by setting bit 1 (RMT) of parameter No.11485 to 1.

Measurement of thread groove

- Measurement of thread groove in arbitrary speed threading mode

1.) Change to arbitrary speed threading mode by specifying M code on condition that the spindle is stopped (S0).
2.) In manual operation, touch the tool to a groove of thread.
3.) In order to measure the groove of thread, groove of thread measurement signal GTMSR <Gn549.4> is set to “1”. Groove of thread measurement signal GTMSR <Gn549.4> is set to “0” after groove of thread measurement completion signal GTMC <Fn546.4> becomes “1”. Machine coordinates of each axis are stored in parameter No.11496 by measuring the groove of thread.
4.) Retract the tool.
5.) Cancel arbitrary speed threading by specifying M code.

> ⚠ **CAUTION**
> 1 In changing to arbitrary speed threading mode, the spindle rotates at a speed of parameter No.4504 to No.4506, and reference position establishment is performed.
> 2 Spindle should not be rotated from the end of reference position establishment to the end of measurement of thread groove.

- Measurement of thread groove with reference position return in Cs contour control mode

In case that a spindle used in re-machining thread is in Cs contour control mode (Cs contour control change completion signal FSCSL <Fn044.1> or Cs contour control change completion signals in each axis FCSSx <Fn274.0 to 3 = “1”) and reference position return is completed (reference position return end signals ZPx<Fn094.0 to 8>=”1”), measuring thread groove is possible. By this way, measuring thread groove can be performed only by operating signals without executing automatic operation.

1) Change to Cs contour control mode on condition that the spindle used in re-machining thread is stopped (S0), and reference position return must be performed. At this time, feedback pulse from this spindle position coder must be selected during measurement of thread groove.
2) In manual operation, touch the tool to a groove of thread.
3) In order to measure the groove of thread, groove of thread measurement signal GTMSR <Gn549.4> is set to “1”. Groove of thread measurement signal GTMSR <Gn549.4> is set to “0” after groove of thread measurement completion signal GTMC <Fn546.4> becomes “1”. Machine coordinates of each axis are stored in parameter No.11496 by measuring the groove of thread.
4) Retract the tool.
5) Cancel Cs contour control mode

> ⚠ **CAUTION**
> 1 The spindle whose feedback pulse from position coder is effective is recognized as a spindle used in re-machining thread. So if another spindle is selected by mistake, groove of thread measurement error (groove of thread measurement error signal GTME<Fn546.5>=”1”) occurs or thread cutting is not performed correctly, so be careful to select the proper spindle.
> 2 If the spindle whose feedback pulse from position coder is effective is not in the state of reference position return end, groove of thread measurement error (groove of thread measurement error signal GTME<Fn546.5>=”1”) occurs.

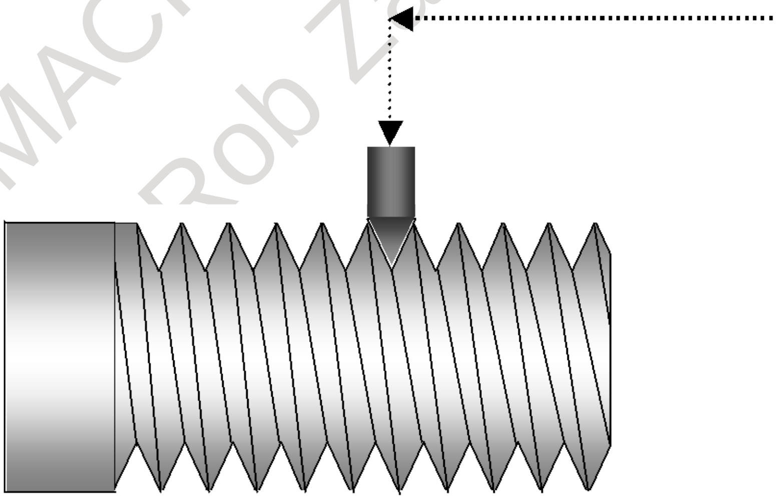


Fig. 6.5.7.2 (a) measurement of thread groove

> ⚠ **CAUTION**
> 1 The groove of thread must be measured when axes except for PMC axis, and spindle for arbitrary speed threading are stopped.
> 2 The groove of thread must be measured with the same tool which is used for re-machining thread. Besides, tool offset should not be changed from measurement of thread groove to re-machining.

> **NOTE**
> The groove of thread cannot be measured in servo-off state.

Re-machining thread

Re-machining thread signal GTMC<Gn549.5> is set to “1”, then arbitrary speed threading is specified. If the re-machining is finished, re-machining thread signal GTMC<Gn549.5> is set to “0”.

<Re-machining thread signal GTMC<Gn549.5> is set to “1”>

```
S_;                     : specifying the spindle speed
Mxx;                    : start of arbitrary speed threading
N1G92 X151.0Z-200.F10.;
N2X150.0;
N3X149.0;
N4X148.0;
G00;
Myy;                    : cancel of arbitrary speed threading
```

<Re-machining thread signal GTMC<Gn549.5> is set to “0”>

> **NOTE**
> 1 If arbitrary speed threading is executed with re-machining thread signal GTMC<Gn549.5> setting to “1”, the timing of threading start is changed according to the measured data of thread groove. Therefore, re-machining thread signal GTMC<Gn549.5> should be set to “0” while re-machining thread is not executed.
> 2 If re-machining thread is executed without measuring the groove of thread after power-on, the alarm PS0532 ”RE-MACHINING OF THREAD CUTTING IMPOSSIBLE” is issued.

- Available threading commands for re-machining thread

Threading commands which are available for re-machining are as follows.

Table 6.5.7.2 (a) Available threading commands for re-machining thread

	Machining center system	Lathe system		
		G code system		
		A	B	C
Threading	G33	G32	G33	G33
Threading cycle	−	G92	G78	G21
Multiple threading cycle	−	G76	G76	G78

Note

- Measurement of thread groove

Measurement position must be the groove except for chamfering part in the case of threading cycle, or the first block groove in the case of continuous threading.

- Re-machining thread

The workpiece which is machined by usual threading can be re-machined. Since re-machining is controlled in arbitrary speed threading mode, re-machined thread may not have the same thread shape completely due to the difference of acceleration/deceleration method. The difference of thread shape may occur at chamfering part of threading cycle or taper part for chamfering of continuous threading.

- Re-machining thread signal

If arbitrary speed threading is executed with re-machining thread signal GTMC<Gn549.5> setting to “1”, the timing of threading start is changed according to the measured data of thread groove. Therefore, re-machining thread signal GTMC<Gn549.5> should be set to “0” while re-machining thread is not executed.

- Command for shifting the threading start angle by address Q

Whether the command for shifting the threading start angle by address Q is enabled or not is selected according to bit 0 (ADQ) of parameter No.11486.

- Bit 0 (ADQ) of parameter No.11486 is set to 0
  The command for shifting the threading start angle by address Q is disabled for re-machining thread. The tool passes the measured groove of thread regardless of Q command.

- Bit 0 (ADQ) of parameter No.11486 is set to 1
  The command for shifting the threading start angle by address Q is enabled for re-machining thread. The measured position of thread groove is assumed to be standard angle (Q=0), and the shift of the threading start angle commanded by address Q is added.

⚠ **CAUTION**

- When bit 0 (ADQ) of parameter No.11486 is set to 1:

  In re-machining the thread which is machined the following program, the groove of thread which is machined by address Q=0 must be measured.

  Example)

  ```
  G32 Z-200.0 F10.0 Q0 ;
        :
  G32 Z-200.0 F10.0 Q90000 ;
        :
  ```

  If the groove of thread which is machined by address Q=90000 is measured, and re-machining is executed by above program, thread is re-machined as shown in Fig. 6.5.7.2 (b).

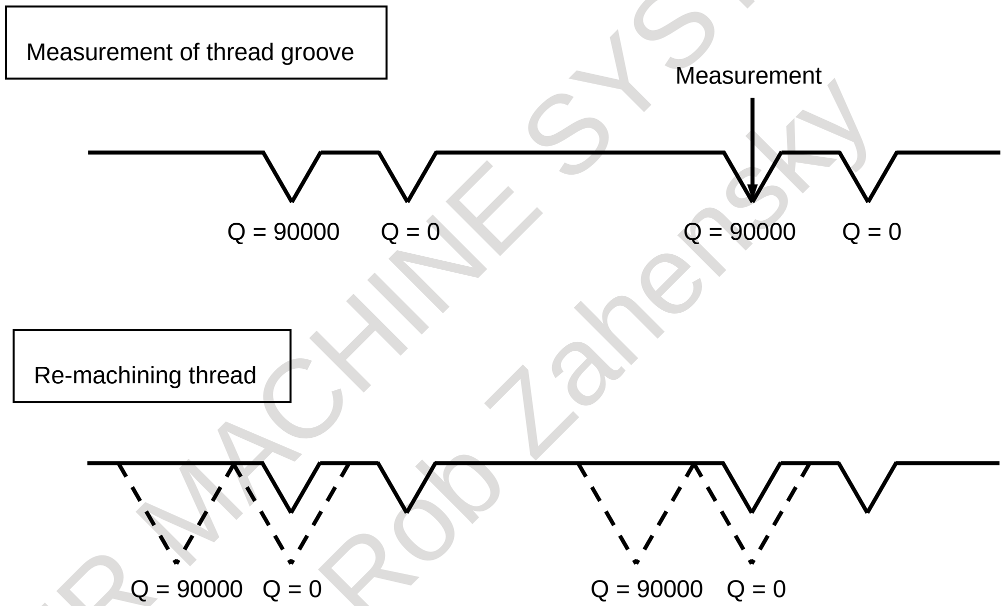


Fig. 6.5.7.2 (b) Example of re-machining thread in bit 0 (ADQ) of parameter No.11486 is set to 1.

Limitation

- Multiple threading cycle

Cutting method of multiple threading cycle is usually one-edge threading with constant cutting amount as shown in Fig. 6.5.7.2 (c) (1). Re-machining of multiple threading cycle is executed so that the groove of thread would be machined at the stored position by measurement as shown in Fig. 6.5.7.2 (c) (2).

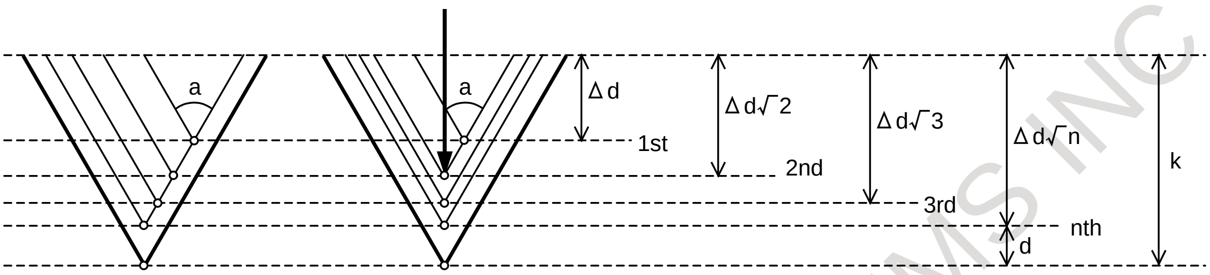


Fig. 6.5.7.2 (c) cutting method of multiple threading cycle re-machining

In the case of memory operation using Series 15 format, even if the following cutting method is selected, re-machining of multiple threading cycle is executed as shown in Fig. 6.5.7.2 (c) (2), too.

P1 : one-edge threading with constant cutting amount
P2 : both-edge zigzag threading with constant cutting amount
P3 : one-edge threading with constant depth of cut
P4 : both-edge zigzag threading with constant depth of cut

- Mirror image

Thread cannot be re-machined with mirror image applied.

Signal

1st spindle speed override signal SOV0 to SOV7<Gn030>
2nd spindle speed override signal SOV20 to SOV27<Gn376>
3rd spindle speed override signal SOV30 to SOV37<Gn377>
4th spindle speed override signal SOV40 to SOV47<Gn378>

[Classification] Input signal

[Function] If bit 3 (MSC) of parameter No.3713 = 1, and bit 4 (EOV) of parameter No.3713 = 1, an override of 0% to 254% of the rotation speed specified by the CNC can be applied to individual spindles separately.

[Operation] Specify an override value, using an 8-bit binary number.
If all bits are “1”, however, an override of 0% is assumed.

> **NOTE**
> If both bit 3 (MSC) and bit 4 (EOV) of parameter No.3713 are not 1, the override signals SOV0 to SOV7 are applied to all spindles that receive speed commands.
> The signals SOV20 to SOV27, SOV30 to SOV37, and SOV40 to SOV47 are invalid.

Cs contour control high speed switching signals
CONH1 to CONH4<Gn549.0 to Gn549.3>

[Classification] Input signal

[Function] These signals specify high speed switching from spindle rotation control to Cs contour control.
When these signals are set to “1”, setting the Cs contour control change signal CON <Gn027.7> (or the Cs contour control change signal in each axis CONSx <Gn274.0 to Gn274.3>) from “0” to “1” starts Cs contour control high speed switching.
When these signals are set to “0”, setting CON (or CONS) from “0” to “1” starts ordinary switching to Cs contour control.
Setting CON (or CONS) from “1” to “0” switches to spindle rotation control. The type of switching is the same as of switching to Cs contour control.
These signals are valid when bit 0 (AST) of parameter No.11485 is set to 1.

Cs contour control high speed switching completion signals
CSMC1 to CSMC4<Fn546.0 to Fn546.3>

[Classification] Output signal

[Function] These signals notify the PMC that the system is put in the Cs contour control mode by Cs contour control high speed switching.

[Output cond.] These signals become “1” under the following condition:

- The system is put in the Cs contour control mode by Cs contour control high speed switching.

These signals become “0” under the following condition:

- The system is put in the Cs contour control mode by ordinary switching to Cs contour control.
- The system is not in the Cs contour control mode.

Forward rotation / Reverse rotation command signal (serial spindle)
SFRA/SRVA<Gn070.5/Gn070.4> : First spindle
SFRB/SRVB<Gn074.5/Gn074.4> : Second spindle
SFRC/SRVC<Gn204.5/Gn204.4> : Third spindle
SFRD/SRVD<Gn266.5/Gn266.4> : Fourth spindle

[Classification] Input signal

[Function] These signals switch between on and off of motor excitation in Cs contour control.

[Operation] When this function is used, in Cs contour control, keep SFRx or SRVx to “1” during:
Mode switching from spindle rotation control to Cs contour control
Mode switching from Cs contour control to spindle rotation control

> **NOTE**
> When SFRx (or SRVx) is set to “0” during mode switching from spindle rotation control to Cs contour control, the spindle decelerates, then reference position establishment is performed.

Reference position establishment starting signal (serial spindle)
CSYCA<Gn304.3> : First spindle
CSYCB<Gn308.3> : Second spindle
CSYCC<Gn312.3> : Third spindle
CSYCD<Gn316.3> : Fourth spindle

[Classification] Input signal

[Function] These signals establish the reference position while the spindle is rotating.

[Operation] Setting these signals to “0” after the reference position establishment starting signal CSYCx is set to “1” starts to establish the reference position while the spindle is rotating.

Reference position establishment completion signal (serial spindle)
CSYFNA<Fn047.7> : First spindle
CSYFNB<Fn051.7> : Second spindle
CSYFNC<Fn170.7> : Third spindle
CSYFND<Fn268.7> : Fourth spindle

[Classification] Output signal

[Function] These signals indicate that reference position establishment while the spindle is rotating is completed.

[Output cond.] These signals become "1" under the following condition:
- Reference position establishment while the spindle is rotating is completed.

These signals become "0" under the following condition:
- Reference position establishment while the spindle is rotating is not completed.
- The system is in the spindle rotation control mode.

One-rotation signal detection status signal for Cs contour control (serial spindle)
CS1DTA<Fn047.6> : First spindle
CS1DTB<Fn051.6> : Second spindle
CS1DTC<Fn170.6> : Third spindle
CS1DTD<Fn268.6> : Fourth spindle

[Classification] Output signal

[Function] These signals indicate that the one-rotation signal has been detected in Cs contour control.

[Output cond.] These signals become "1" under the following condition:
- The one-rotation signal has been detected in Cs contour control.

These signals become "0" under the following condition:
- The one-rotation signal has not been detected in Cs contour control.

SV reverse signals SVRVS1 to SVRVS8<Gn523>
[Classification] Input signal

[Function] These signals cause the respective axes to reverse their rotation direction in the arbitrary speed threading mode.

[Operation] Setting these signals to "1" cause the respective axes to reverse their rotation direction. Resetting these signals to "0" cause the respective axes to resume their normal rotation direction.

> **NOTE**
> In this function, because spindle rotation control mode is switched to arbitrary speed threading mode (position control) immediately while the spindle is rotating, rotation direction in arbitrary speed threading might be switched from that in spindle rotation control due to machine configuration or parameter setting. For this case, SV reverse signal SVRVS1 to SVRVS8 <Gn523> is set to "1" before changing to arbitrary speed threading mode so that the rotation direction in arbitrary speed threading would be the same as that in spindle rotation control. Besides, SV reverse signal SVRVS1 to SVRVS8 <Gn523> is set to "0" after canceling the arbitrary speed threading mode.

Groove of thread measurement signal GTMSR<Gn549.4>
[Classification] Input signal

[Function] The groove of thread is measured.

[Operation] When groove of thread measurement signal is set to “1”, the groove of thread is measured.

Groove of thread measurement completion signal GTMC<Fn546.4>

[Classification] Output signal

[Function] Completion of thread groove measurement is indicated.

[Output cond.] This signal becomes “1” when:

- Groove of thread measurement signal is set to “1”, then the measurement of thread groove is completed.

This signal becomes “0” when:

- Groove of thread measurement signal is set from “1” to “0”.

Groove of thread measurement error signal GTME<Fn546.5>

[Classification] Output signal

[Function] It indicates that error is detected while measurement of thread groove.

[Output cond.] This signal becomes “1” when:

- The groove of thread is measured when axes except for PMC axis, and spindle (for arbitrary speed threading) are not stopped. This signal becomes “1” when the groove of thread measurement completion signal becomes “1” at the same time. This signal becomes “0” when:
- Groove of thread measurement signal is set from “1” to “0”.

Sequence of thread groove measurement

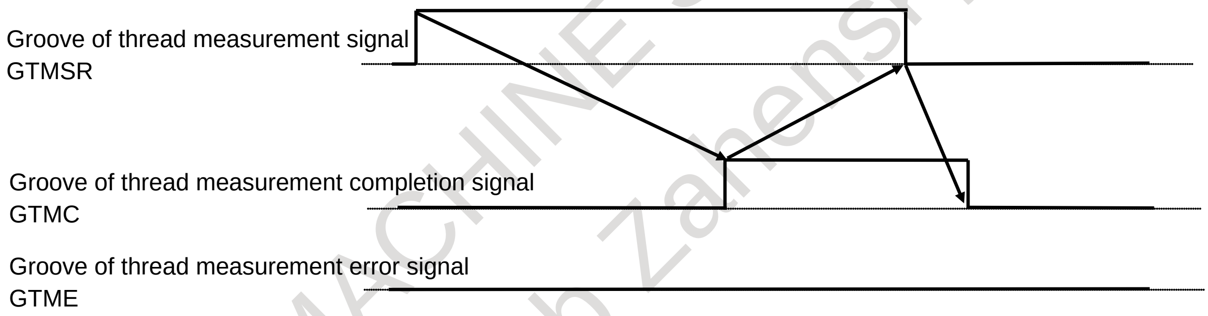


Fig. 6.5.7.2 (d) completion of thread groove measurement

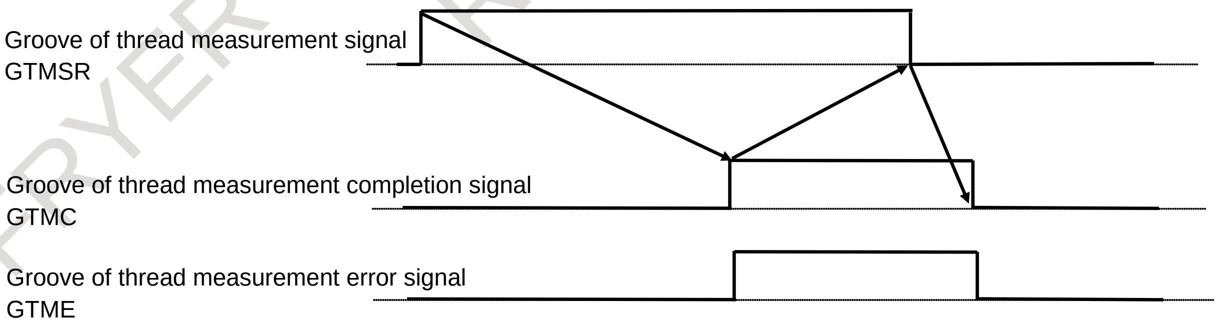


Fig. 6.5.7.2 (e) error of thread groove measurement

Re-machining thread signal RMTC<Gn549.5>

[Classification] Input signal

[Function] Re-machining thread is executed.

[Operation] In order to re-machine the thread, arbitrary speed threading is executed with setting this signal to “1”.

If re-machining thread is executed without measuring the groove of thread after power-on, the alarm PS0532 “RE-MACHINING OF THREAD CUTTING IMPOSSIBLE” is issued. This signal may be set to “1” in arbitrary speed threading. However, in this case, this signal must be set to “1” while the buffering is prevented by specifying M code of preventing buffering.

Chamfering for arbitrary speed threading signal ASTC<Gn549.6>

[Classification] Input signal

[Function] In arbitrary speed threading, the parameter is selected to use with chamfering value and angle in thread cutting cycles.

[Operation] When ASTC is “0”, chamfering value and angle in usual thread cutting cycles (Parameters No.5130, No.5131) are used.
When ASTC is “1“, chamfering value and angle in thread cutting cycles for arbitrary speed threading (Parameters No.11497, No.11498) are used.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn030	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0

	#7	#6	#5	#4	#3	#2	#1	#0
Gn376	SOV27	SOV26	SOV25	SOV24	SOV23	SOV22	SOV21	SOV20

	#7	#6	#5	#4	#3	#2	#1	#0
Gn377	SOV37	SOV36	SOV35	SOV34	SOV33	SOV32	SOV31	SOV30

	#7	#6	#5	#4	#3	#2	#1	#0
Gn378	SOV47	SOV46	SOV45	SOV44	SOV43	SOV42	SOV41	SOV40

	#7	#6	#5	#4	#3	#2	#1	#0
Gn523	SVRVS8	SVRVS7	SVRVS6	SVRVS5	SVRVS4	SVRVS3	SVRVS2	SVRVS1

	#7	#6	#5	#4	#3	#2	#1	#0
Gn549		ASTC	RMTC	GTMSR	CONH4	CONH3	CONH2	CONH1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn546			GTME	GTMC	CSMC4	CSMC3	CSMC2	CSMC1

- For first serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn070			SFRA	SRVA				

	#7	#6	#5	#4	#3	#2	#1	#0
Gn304					CSYCA			

	#7	#6	#5	#4	#3	#2	#1	#0
Fn047	CSYFNA	CS1DTA						

- For second serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn074			SFRB	SRVB				

	#7	#6	#5	#4	#3	#2	#1	#0
Gn308					CSYCB			

	#7	#6	#5	#4	#3	#2	#1	#0
Fn051	CSYFNB	CS1DTB						

- For third serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn204			SFRC	SRVC				

	#7	#6	#5	#4	#3	#2	#1	#0
Gn312					CSYCC			

	#7	#6	#5	#4	#3	#2	#1	#0
Fn170	CSYFNC	CS1DTC						

- For fourth serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn266			SFRD	SRVD				

	#7	#6	#5	#4	#3	#2	#1	#0
Gn316					CSYCD			

	#7	#6	#5	#4	#3	#2	#1	#0
Fn268	CSYFND	CS1DTD						

Parameter

Parameters which are used for this function are as follows.

	#7	#6	#5	#4	#3	#2	#1	#0
11485							RMT	AST

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 AST Arbitrary speed threading is:
0: Disabled.
1: Enabled.

#1 RMT Re-machining thread is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
11486						AMM	ART	ADQ

[Input type] Parameter input
[Data type] Bit path

#0 ADQ Command for shifting the threading start angle by address Q in re-machining thread is:
0: Disabled.
1: Enabled.

#1 ART Arbitrary speed threading mode is:

0: Canceled by reset.
1: Not canceled by reset.

NOTE
1 In case of this parameter is 0, if reset operation is executed in arbitrary speed threading mode, Cs contour control change signal CON <Gn027.7> or Cs contour control change signals in each axis CONS <Gn274.0 to 3> must be set to “0”.
2 In case of this parameter is 1, if reset operation is executed in arbitrary speed threading mode, Cs contour control change signal CON <Gn027.7> or Cs contour control change signals in each axis CONS <Gn274.0 to 3> must be maintained to “1”. If it cancels the arbitrary speed threading mode, M code of cancel the arbitrary speed threading (No.11488) must be specified and Cs contour control change signal CON <Gn027.7> or Cs contour control change signals in each axis CONS <Gn274.0 to 3> must be set to “0”.

#2 AMM In arbitrary speed threading mode, M code to start arbitrary speed threading mode is
0: Disable to command (Alarm (PS0529) "THREADING COMMAND IMPOSSIBLE" occurs).
1: Enabled to command.

NOTE
In case of this parameter is 1, M code to start arbitrary speed threading can be re-commanded in arbitrary speed threading mode. Arbitrary speed threading mode is already selected. So process to change to arbitrary speed threading is not performed, but M code is output. Please operate properly with PMC ladder.

11487	M code to start arbitrary speed threading mode

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] None
[Valid data range] 0 to 99999999
This parameter sets the M code to start arbitrary speed threading mode.

NOTE
1 The parameter setting must not be the same as the M code used for any other function.
2 When this parameter is set to 0, this function is invalid.
3 If the same value is set for two or more Cs contour control axes, the alarm PS0531, “THREADING PARAMETER ERROR” is issued.
4 The M code set in this parameter prevents buffering.

11488	M code to cancel arbitrary speed threading mode

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] None
[Valid data range] 0 to 99999999

This parameter sets the M code to cancel arbitrary speed threading mode.

> **NOTE**
> 1 The parameter setting must not be the same as the M code used for any other function.
> 2 When this parameter is set to 0, this function is invalid.
> 3 If the same value is set for two or more Cs contour control axes, the alarm PS0531, "THREADING PARAMETER ERROR" is issued.
> 4 The M code set in this parameter prevents buffering.

11489	Acceleration in arbitrary speed threading

[Input type] Parameter input
[Data type] Real axes
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, 0.0 to +10000.0.)
This parameter sets acceleration of major axis in arbitrary speed threading.
If this parameter is set to 0, acceleration is assumed to 100000.0.

11490	Spindle speed arrival level in arbitrary speed threading

[Input type] Parameter input
[Data type] word path
[Unit of data] min^{-1}
[Valid data range] 0 to 32767
Threading is started if the spindle speed is arrived within the level set in this parameter.

> **NOTE**
> Threading is not started if the spindle speed is not arrived within the level set in this parameter.

11496	Measurement result of thread groove

[Input type] Parameter input
[Data type] Real axis

[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Measurement result of thread groove is stored.

> **NOTE**
> This parameter stores measurement result of thread groove. Don't input this parameter.

11497	Chamfering value in thread cutting cycles G92 and G76 for arbitrary speed threading

[Input type] Parameter input

[Data type] Byte path
[Unit of data] 0.1
[Valid data range] 0 to 127

This parameter sets a chamfering value in the thread cutting cycle (G76) of a multiple repetitive canned cycle and in the thread cutting cycle (G92) of a canned cycle for arbitrary speed threading.
Let L be a lead. Then, a chamfering value range from 0.1L to 12.7L is allowed.
To specify a chamfering value of 10.0L, for example, specify 100 in this parameter.

11498	Chamfering angle in thread cutting cycles G92 and G76 for arbitrary speed threading

[Input type] Parameter input
[Data type] Byte path
[Unit of data] deg
[Valid data range] 1 to 89

This parameter sets a thread chamfering angle in a thread cutting cycle (G92/G76) for arbitrary speed threading. When 0 is set, an angle of 45 degrees is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
11001		DCS						

[Input type] Parameter input
[Data type] Bit axis

#6 DCS Special acceleration for deceleration is:
0: Disabled.
1: Enabled.

11020	Acceleration/deceleration switching speed (S_0) for each axis
11021	Acceleration/deceleration switching speed (S_1) for each axis
25700	Acceleration/deceleration switching speed (S_{10}) for deceleration
25701	Acceleration/deceleration switching speed (S_{11}) for deceleration

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

These parameter set the speed at which acceleration/deceleration is changed for each axis (S_0, S_1, S_{10}, S_{11}).

NOTE
If this parameter is set the following values, the subsequent acceleration/deceleration is disabled.
- Over the maximum speed
- 0
- The value of upper step is smaller than that of lower step

11030	Individual acceleration/deceleration 1 (0 to S_0)
11031	Individual acceleration/deceleration 2 (S_0 to S_1)

11032	Individual acceleration/deceleration 3 (S_1 to maximum speed)
25710	Special acceleration for deceleration (0 to S_{10})
25711	Special acceleration for deceleration (S_{10} to S_{11})
25712	Special acceleration for deceleration (S_{11} to maximum speed)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}/s
[Valid data range] 0 to 99999999

This parameters set acceleration/deceleration of sections 0 to S_0, S_0 to S_1, $S_1$to maximum speed, 0 to S_{10}, S_{10} to S_{11}, S_{11} to maximum speed for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx
#1 ROSx Setting linear or rotary axis.

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No.3624)
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No.1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No.3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type. (Refer to parameter No.3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

	#7	#6	#5	#4	#3	#2	#1	#0
1008						RRLx		ROAx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROAx The rotary axis roll-over is
0: Invalid
1: Valid

> **NOTE**
> ROAx specifies the function only for a rotary axis (for which bit 0 (ROTx) of parameter No.1006 is set to 1)

#2 RRLx Relative coordinates are
0: Not rounded by the amount of the shift per one rotation
1: Rounded by the amount of the shift per one rotation

> **NOTE**
> 1 RRLx is valid only when ROAx is 1.
> 2 Assign the amount of the shift per one rotation in parameter No.1260.

	#7	#6	#5	#4	#3	#2	#1	#0
1408					IRCx			

[Input type] Parameter input
[Data type] Bit axis

#3 IRCx The least input increment of the maximum cutting feedrates set in parameter No.1430 and 1432 and the maximum servo motor speed set in parameter No.12255 is:
0: Not multiplied by ten.
1: Multiplied by ten.

Set this parameter for the following axes, which are operated by the following functions:
- Spindle control axis by servo motor
- Tool rotary axis in the polygon turning function

The maximum cutting feedrates parameter (No.1430/No.1432) is not checked in the undermentioned function.
When this parameter is set, only the least input increment of the servo motor maximum speed parameter (No.12255) is multiplied by ten.
- Arbitrary speed threading

If a rotation speed of 1000 (1/min) (=360000 (deg/min)) is to be used when this parameter is set to 1, set 36000.0 in parameter No. 1430/1432.

1430	Maximum cutting feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Specify the maximum cutting feedrate for each axis.

1825	Servo loop gain for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01/sec
[Valid data range] 1 to 32767

Set the loop gain for position control for each axis.
When the machine performs linear and circular interpolation (cutting), the same value must be set for all axes. When the machine requires positioning only, the values set for the axes may differ from one another. As the loop gain increases, the response by position control is improved. A too large loop gain, however, makes the servo system unstable. The relationship between the positioning deviation (the number of pulses counted by the error counter) and the feedrate is expressed as follows:

Positioning deviation = Feedrate / (60 * Loop gain)
Unit : Positioning deviation mm, inch or deg
Feedrate mm/min, inch/min, or deg/min
Loop gain 1/sec

	#7	#6	#5	#4	#3	#2	#1	#0
1610			THLx	JGLx			CTBx	CTLx

[Input type] Parameter input
[Data type] Bit axis

#0 CTLx Acceleration/deceleration in cutting feed or dry run during cutting feed
0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied.

NOTE
This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

#1 CTBx Acceleration/deceleration in cutting feed or dry run during cutting feed
0: Exponential acceleration/deceleration or linear acceleration/deceleration is applied. (depending on the setting in bit 0 (CTLx) of parameter No.1610)
1: Bell-shaped acceleration/deceleration is applied.

NOTE
This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

#4 JGLx Acceleration/deceleration in jog feed
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied.
(Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No.1610)

> **NOTE**
> This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

#5 THLx Acceleration/deceleration in threading cycles
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied.
(Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No.1610)
As the time constant and FL rate, however, the settings of parameters Nos.1626 and 1627 for threading cycles are used.

	#7	#6	#5	#4	#3	#2	#1	#0
1611								CFR

[Input type] Parameter input
[Data type] Bit path

#0 CFR For retraction after threading in the threading cycles G92 and G76:
0: The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No.1626) and FL rate (parameter No.1627).
1: The type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant.

> **NOTE**
> If this parameter is set to 1, a check is made before a retraction to see that the specified feedrate has become 0 (the delay in acceleration/deceleration has become 0). For retraction, the rapid traverse rate (parameter No.1420) is used, regardless of the setting of parameter No.1466. When this parameter is set to 0, parameter No.1466 is used as the feedrate for retraction. As acceleration/deceleration used for retraction, only acceleration/deceleration after interpolation is used. Rapid traverse acceleration/deceleration before interpolation and optimum torque acceleration/deceleration are disabled.

1626	Acceleration/deceleration time constant in threading cycles for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
Set a time constant for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis.

1627	FL rate for acceleration/deceleration in threading cycles for each axis

[Input type] Parameter input
[Data type] Real axis

[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set an FL rate for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis. Set 0 at all times except in a special case.

	#7	#6	#5	#4	#3	#2	#1	#0
4002				CSDRCT				

[Input type] Parameter input
[Data type] Bit spindle

#4 CSDRCT In Cs contour control, the rotation direction signal (SFR/SRV) function is:
0: Used.
1: Not used.
When using this function, setting to 1 (the rotation direction signal function is not used) is recommended.
When this parameter is set to 0, keep SFRx or SRVx to “1” during the following operation.
- Mode switching from spindle rotation control to Cs contour control
- Mode switching from Cs contour control to spindle rotation control
- Arbitrary speed threading
- Measurement of thread groove
- Re-machining thread

4069	Position gain in Cs contour control (HIGH gear)

4070	Position gain in Cs contour control (MEDIUM HIGH gear)

4071	Position gain in Cs contour control (MEDIUM LOW gear)

4072	Position gain in Cs contour control (LOW gear)

[Input type] Parameter input
[Data type] Integral word spindle
[Unit of data] 0.01/sec
[Valid data range] 0 to 32767
Set the position gain in Cs contour control.

4504	Cs contour control high speed switching: Reference position establishment speed

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 1000
This parameter sets the spindle speed during reference position establishment.

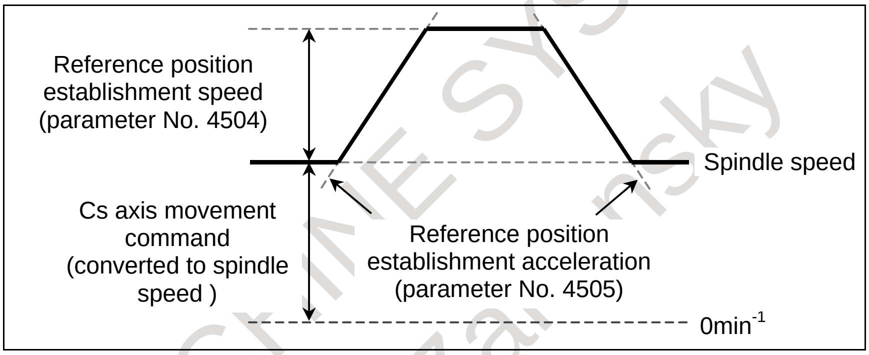
NOTE

Set a value in this parameter so that the absolute value of "Cs axis move command (converted to the spindle speed) ± setting" does not exceed the maximum spindle speed in Cs contour control (set in parameter No.4021). If the value exceeds the maximum spindle speed, a symptom including the following may occur:
- Reference position establishment is not completed.
- Positional deviation increases and a large acceleration/deceleration shock occurs.

4505	Cs contour control high speed switching: Reference position establishment acceleration

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] min^{-1}/sec
[Valid data range] 3 to 32767

This parameter sets the acceleration rate applied to the spindle during reference position establishment.



NOTE

Set a value so that the spindle acceleration/deceleration capacity is not exceeded. If the capacity is exceeded, positional deviation may increase, a large acceleration/deceleration may occur, or another symptom may occur.

4506	Cs contour control high speed switching: Bell-shaped acceleration/deceleration time constant for the reference position establishment

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] msec
[Valid data range] 8 to 64

This parameter sets the bell-shaped acceleration/deceleration time constant for reference position establishment. If a value outside the valid data range is input, the setting is clamped to the upper or lower limit.

4507	Cs contour control high speed switching: Reference position establishment completion level

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] detection unit
[Valid data range] 0 to 32767

This parameter sets the completion level for reference position establishment. When the absolute value of spindle positional deviation becomes within the setting after the move command for reference position establishment has been distributed, the reference position establishment completion signal CSYFNx becomes 1. When this parameter is set to 0, CSYFNx becomes 1 immediately after the move command for reference position establishment has been distributed.

Alarm and message

Number	Message	Description
PS0528	THREADING FORMAT ERROR	The format of arbitrary speed threading is invalid.
PS0529	THREADING COMMAND IMPOSSIBLE	1) The following commands cannot be specified in arbitrary speed threading mode. - Threading command except for threading (G32), threading cycle (G92), and multiple threading cycle (G76). - Taper angle is smaller than that of the last block in the case of continuous threading. Chamfering angle is smaller than the taper angle in the case of threading cycle. - Infeed axis cannot be decelerated to 0 with acceleration which is calculated automatically in continuous threading. - Synchronous control, composite control, and superimposed control - Manual reference position return - Spindle command synchronous control - Simple spindle electronic gear box 2) Arbitrary speed threading is specified in Cs contour control. 3) Spindle software does not support arbitrary speed threading. Cs contour control change signal is not set to “0” after resetting CNC in arbitrary speed threading mode.
PS0530	EXCESS VELOCITY IN THREADING	Feedrate exceeds the maximum cutting feedrate.
PS0531	THREADING PARAMETER ERROR	Parameter setting for arbitrary speed threading is invalid. Possible causes are: - An M code value to start arbitrary speed threading is used for multiple Cs contour control axes. - An M code value to cancel arbitrary speed threading is used for multiple Cs contour control axes. - Acceleration of feed axis (parameter No.11489) is smaller than that of spindle (Cs contour control axis) (parameter No.11030 to No.11032, No.25710 to No.25712).

Number	Message	Description
DS0083	THREADING SIGNAL ERROR	Signal setting is invalid. Possible causes are: - Reference position establishment starting signal CSYC is set to “1” before the Cs contour control high speed switching completion signal CSMCx becomes “1”. - FIN for start of arbitrary speed threading is returned before the reference position establishment completion signal becomes “1”. - FIN for cancel of arbitrary speed threading is returned before the Cs contour control high speed switching completion signal CSMCx becomes “0”. - Spindle which is commanded M code for arbitrary speed threading is not specified to get the feedback pulse from position coder. - State of signal (the Cs contour control high speed switching completion signal CSMCx and the reference position establishment completion signal) is not corresponding to arbitrary speed threading mode.
PS0532	RE-MACHINING OF THREAD CUTTING IMPOSSIBLE	Re-machining thread cannot be executed. Possible causes are: - Groove of thread is not measured. - Measured data is outside of threading path. - Mirror image is applied to the first axis on the plane or the second axis on the plane.

6.5.8 Rapid Traverse Overlap in Threading Cycle

T

Overview

In threading cycle of canned cycle, a cycle time can be shortened with rapid traverse overlap between blocks.

> **NOTE**
> This function is invalid in “Multiple repetitive cycle”.

Explanation

To use this function, set the following parameters.

- Bit 2 (TOD) of parameter No.1612 = 1
  Rapid traverse overlap is valid in threading cycle.
- Bit 0 (CFR) of parameter No.1611 = 1
  For retraction after threading, the type of acceleration/deceleration after interpolation for rapid traverse is used.
- Bit 4 (PRT) of parameter No.1603 = 0
  Acceleration/deceleration of acceleration fixed type is used for positioning of linear interpolation type.

> **NOTE**
> 1 It is not necessary to set bit 4 (RTO) of parameter No.1601 for this function.
> 2 On condition that both standard rapid traverse blocks overlap and this function are available, this function is valid in threading cycle. And, standard rapid traverse block overlap is valid in other cases.

Rapid traverse overlap is executed at the following corners.
- Between the retraction of X-axis (3) and the return to the cycle start point of Z-axis (4)
- Between the return to the cycle start point of Z-axis (4) and positioning to the threading start point of X-axis (1)

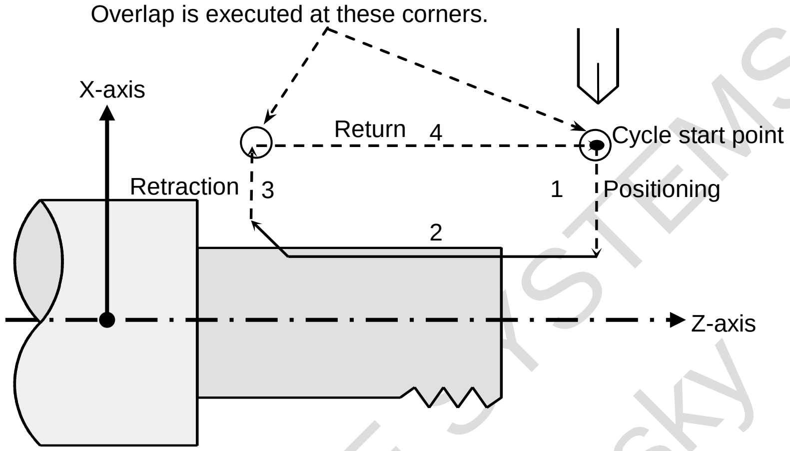


Fig. 6.5.8 (a) Rapid traverse overlap in threading cycle

Overlap is executed as the following figure.
After interpolation is completed, the next block starts when the delay by acceleration/deceleration control is the distance of overlap or below.

Example : Overlap between retraction and return

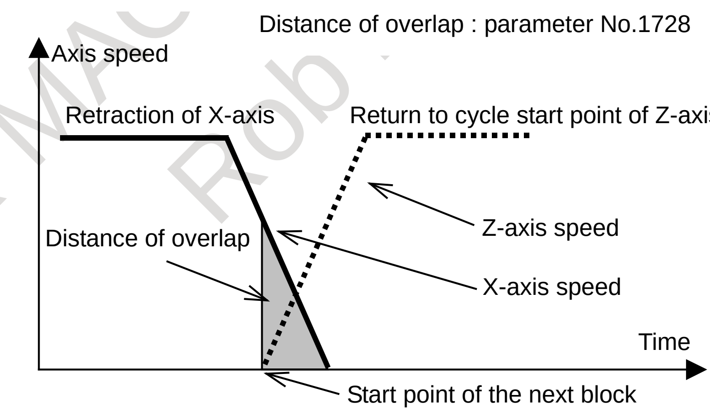


Fig. 6.5.8 (b) Overlap between retraction and return

Distances of overlap are specified with the following parameters.
- Parameter No.1728
  Distance of overlap between retraction (3) and return (4) in threading cycle
- Parameter No.1729
  Distance of overlap between return (4) and positioning of the next threading cycle (1)

Threading cycle retract can be used for this function.

Restrictions

This function is invalid in the following cases.

- Acceleration/deceleration before raid traverse interpolation is used.
- The bit 6 (RDR) of parameter No.1401 = 1 (dry run for rapid traverse is valid), and the dry run signal DRN<Gn046.7> is "1" and the manual rapid traverse selection signal RT<Gn019.7> is set to "0".
- The bit 1 (LRP) of parameter No.1401 = 1 (positioning is performed with linear interpolation) and bit 4 (PRT) of parameter No.1603 = 1 (acceleration/deceleration of time fixed type is used).

Besides, there are the following restrictions.
- The threading cycle must be commanded in G40 mode (Tool nose radius compensation cancel mode).
- Rapid traverse block overlap disable signal ROVLP<Gn053.5> is invalid in this function.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1612						TOD		

[Input type] Parameter input
[Data type] Bit path

#2 TOD Rapid traverse overlap in threading cycle is
0: Invalid.
1: Valid.
In threading cycle of canned cycle, rapid traverse overlap at the following corners.
- Between the retraction of X-axis and the return to the cycle start point of Z-axis
- Between the return to the cycle start of Z-axis and positioning to the threading start point of X-axis

Distances of overlap are specified with parameter No.1728 and No.1729.

1728	Distance of rapid traverse overlap between retraction and return in threading cycle

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B)) (When the increment system is IS-B, 0.0 to +999999.999)
In threading cycle of canned cycle, return to the cycle start point of Z-axis starts when the delay of X-axis by acceleration/deceleration control is the distance of this parameter or below.

NOTE
1 Set radius value in this parameter.
2 This parameter is valid on condition that parameter TOD (bit 2 of No.1612) is "1".

1729	Distance of rapid traverse overlap between return and positioning of the next block in threading cycle

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B)) (When the increment system is IS-B, 0.0 to +999999.999)
In threading cycle of canned cycle, positioning to the threading start point of X-axis starts when the delay of Z-axis by acceleration/deceleration control is the distance of this parameter or below.

> **NOTE**
> 1 Set radius value in this parameter.
> 2 This parameter is valid on condition that parameter TOD (bit 2 of No.1612) is “1”.

6.6 HELICAL INTERPOLATION

Overview

Helical interpolation that moved tool on the helically is enabled by specifying up to two other axes which move synchronously with the circular interpolation by circular commands.
The feedrate command can select whether to specify tangential velocity of an arc or to specify tangential velocity including a helical axis by the setting of bit 5 (HTG) of parameter No.1403.

An F command specifies a feed rate along a circular arc, when HTG is specified to 0. Therefore, the feed rate of the helical axis is as follows:

$$F \times \frac{\text{Length of helical axis}}{\text{Length of circular arc}}$$

Determine the feed rate so the helical axis feed rate does not exceed any of the various limit values.

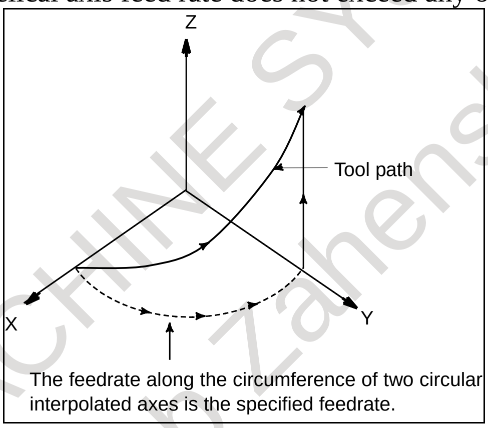


Fig. 6.6 (a)

If HTG is set to 1, specify a feedrate along the tool path including the helical axis. Therefore, the tangential velocity of the arc is expressed as follows:

$$F \times \frac{\text{Length of arc}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of helical axis})^2}}$$

The velocity along the linear axis is expressed as follows:

$$F \times \frac{\text{Length of linear axis}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of helical axis})^2}}$$

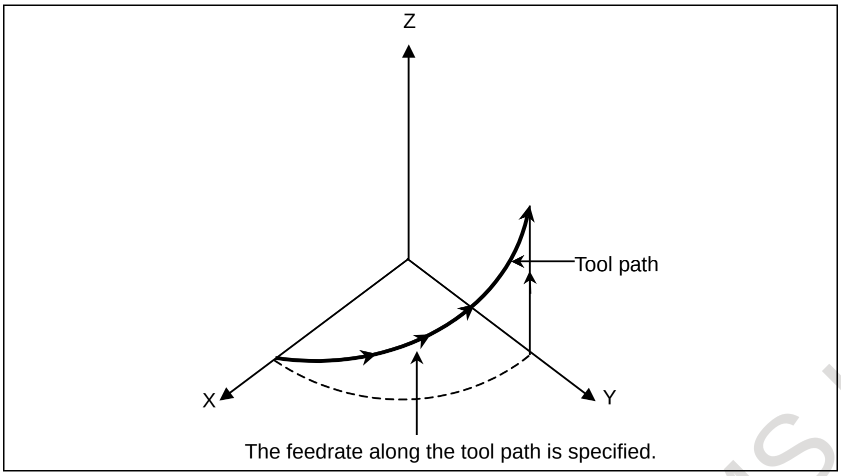


Fig. 6.6 (b)

If three or more axes were specified as helical axes in the helical interpolation mode, alarm (PS0232)"TOO MANY HELICAL AXIS COMMAND" is issued.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1403			HTG					

[Input type] Parameter input
[Data type] Bit path

#5 HTG The feedrate for helical interpolation is:
0: Specified using the feedrate along the tangent to an arc
1: Specified using the feedrate along axes including a helical axis

Alarm and message

Number	Message	Description
PS0232	TOO MANY HELICAL AXIS COMMAND	Three or more axes were specified as helical axes in the helical interpolation mode.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Helical interpolation

6.7 POLAR COORDINATE INTERPOLATION

T

Overview

Polar coordinate interpolation is a function that exercises contour control in converting a command programmed in a Cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece). This function is useful for grinding a cam shaft.

Explanation

G12.1 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane (Fig. 6.7). Polar coordinate interpolation is performed on this plane.

- Polar coordinate interpolation plane

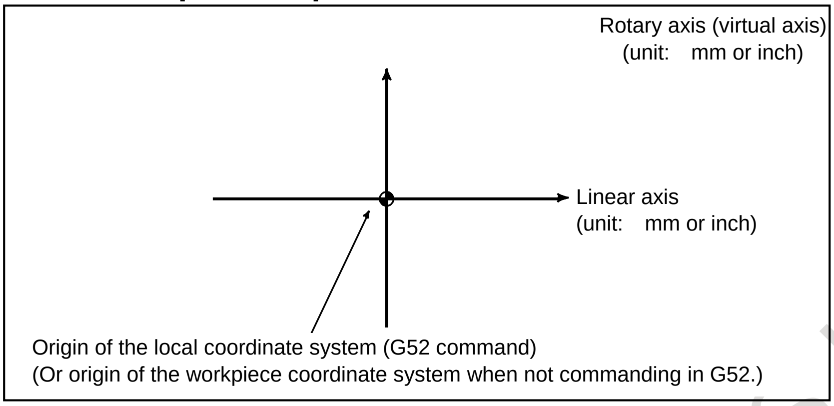


Fig. 6.7 (a) Polar coordinate interpolation plane

When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G13.1).
The linear and rotation axes for polar coordinate interpolation must be set in parameters Nos. 5460 and 5461 beforehand.

> ⚠ **CAUTION**
> The plane used before G12.1 is specified (plane selected by G17, G18, or G19) is canceled. It is restored when G13.1 (canceling polar coordinate interpolation) is specified.
> When the system is reset, polar coordinate interpolation is canceled and the plane specified by G17, G18, or G19 is used.

Parameter

1430	Maximum cutting feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Specify the maximum cutting feedrate for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
5450		PRD	PIC			PLS		PDI

[Input type] Parameter input
[Data type] Bit path

#0 PDI When the second axis on the plane in the polar coordinate interpolation mode is based on radius specification:
0: Radius specification is used.
1: Diameter specification is used.

#2 PLS The polar coordinate interpolation shift function is:
0: Not used.
1: Used.
This enables machining using the workpiece coordinate system with a desired point which is not the center of the rotation axis set as the origin of the coordinate system in polar coordinate interpolation.

#5 PIC The initial display type of the absolute position of the linear and rotary axes to be used in the polar coordinate interpolation is:
0: Actual position.
1: Cartesian coordinates in the polar coordinate interpolation plane.

> **NOTE**
> 1 The display type can be switched via the soft key [CARTES ON] / [CARTES OFF] in the absolute position display screen.
> 2 This parameter does not change if switching the display type via the soft key.

#6 PRD When the first axis on the plane in the polar coordinate interpolation mode is based on diameter specification:
0: Diameter specification is used.
1: Radius specification is used.

5460	Axis (linear axis) specification for polar coordinate interpolation

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes
This parameter sets control axis numbers of linear axis to execute polar interpolation.

5461	Axis (rotation axis) specification for polar coordinate interpolation

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes
This parameter sets control axis numbers of rotation axis to execute polar interpolation.

5463	Automatic override tolerance ratio for polar coordinate interpolation

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 100
Typical setting: 90% (treated as 90% when set to 0)
Set the tolerance ratio of the fastest cutting feedrate to the speed of the rotation axis during automatic override of polar coordinate interpolation.

5464	Compensation for error on hypothetical axis of polar coordinate interpolation

[Input type] Parameter input
[Data type] Byte path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(For IS-B, -999999.999 to +999999.999)
This parameter is used to set the error if the center of the rotation axis on which polar coordinate interpolation is performed is not on the X-axis.
If the setting of the parameter is 0, regular polar coordinate interpolation is performed.

Alarm and message

Number	Message	Description
PS0145	ILLEGAL USE OF G12.1/G13.1	The axis No. of plane selection parameter No. 5460 (linear axis) and No. 5461(axis of rotation) in the polar coordinate interpolation mode is out of range (1 to number of controlled axes).
PS0146	ILLEGAL USE OF G-CODE	The modal G code group contains an illegal G code in the polar coordinate interpolation mode or when a mode was canceled. Only the following G codes are allowed: G40, G50, G69.1 An illegal G code was specified while in the polar coordinate interpolation mode. The following C codes are not allowed: G27, G28, G30, G31 to G31.4, G37 to G37.3, G52, G92, G53, G17 to G19, G81 to G89, G68 In the 01 group, G codes other than G01, G02 and G03 cannot be specified.
DS1512	EXCESS VELOCITY	The feedrate of the linear axis during polar coordinate interpolation exceeded the maximum cutting feedrate.
DS1514	ILLEGAL MOTION IN G12.1 MODE	In a hypothetical axis direction compensation during the polar coordinate interpolation mode, an attempt is made to travel to the area in which the travel cannot be made.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Polar coordinate interpolation

6.8 CYLINDRICAL INTERPOLATION

6.8.1 Cylindrical Interpolation

Overview

The cylindrical interpolation function performs linear interpolation or arc interpolation with another axis by converting the travel distance of a rotation axis specified in degrees to the travel distance on the circumference.
Since programming is allowed with the side of a cylinder developed, a program for cylinder cam grooving or other machining can easily be created.

Explanation

- Plane selection (G17, G18, G19)

To specify a G code for plane selection, set the rotary axis in parameter No. 1022 as a linear axis that is one of the basic three axes of the basic coordinate system or an axis parallel to one of the basic axes. For example, when rotary axis C-axis is assumed to be parallel to the X-axis, specifying G17, axis address C, and Y at the same time can select a plane formed by the C-axis and Y-axis (the Xp-Yp plane).
Only an axis can be set to the rotary axis of cylindrical interpolation.

6.8.2 Cylindrical Interpolation by Plane Distance Command

Overview

In the conventional rotary axis command in cylindrical interpolation, the angle of the rotary axis is specified.
This function enables the rotary axis command in cylindrical interpolation to be specified by distance on the developed plane by setting bit 2(DTO) of parameter No.3454.

NOTE
1. Do not change the parameter DTO while cylindrical interpolation mode.
2. To use the function, enable "cylindrical interpolation" (bit 6 (NCL) of parameter No.8137#6 is 0).

6.8.3 Cylindrical Interpolation Cutting Point Compensation

Overview

The conventional cylindrical interpolation function controls the movement of the tool center so that the tool axis moves along a specified path on the cylindrical surface to always face toward the rotation axis of the workpiece (cylindrical axis).
On the other hand, the cylindrical interpolation cutting point control function controls the tool so that the tangent from the tool to the cutting face of a contour figure passes the rotation center of the workpiece at all times.
That is, the cutting face of a contour figure is always perpendicular to the cylinder. Therefore, the figure of the cutting face can be kept identical regardless of the cutter compensation.

NOTE
To use the function, enable "cylindrical interpolation" (bit 6 (NCL) of parameter No.8137#6 is 0).

Parameter

- **Settings common to cylindrical interpolation, cylindrical interpolation by plane distance command, and cylindrical interpolation cutting point compensation**

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx
#1 ROSx Setting linear or rotary axis.

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No. 1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.

ROSx	ROTx	Meaning
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

1022	Setting of each axis in the basic coordinate system

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

To determine a plane for circular interpolation, cutter compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane) and a 3-dimensional tool compensation space (XpYpZp), specify which of the basic three axes (X, Y, and Z) is used for each control axis, or a parallel axis of which basic axis is used for each control axis.
A basic axis (X, Y, or Z) can be specified only for one control axis.
Two or more control axes can be set as parallel axes for the same basic axis.

Setting	Meaning
0	Rotary axis (Neither the basic three axes nor a parallel axis)
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

In general, the increment system and diameter/radius specification of an axis set as a parallel axis are to be set in the same way as for the basic three axes.

1260	The shift amount per one rotation of a rotary axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the shift amount per one rotation of a rotary axis.
For the rotary axis used for cylindrical interpolation, set the standard value.

- Parameters required for cylindrical interpolation by plane distance command

	#7	#6	#5	#4	#3	#2	#1	#0
3454						**DTO**		

[Input type] Parameter input
[Data type] Bit path

#2 DTO The method of specifying a rotation axis in cylindrical interpolation mode is set.
0: In cylindrical interpolation mode, the rotation axis is specified by angle.
1: In cylindrical interpolation mode, the rotation axis is specified by distance on an expanded plane.

- Parameters required for cylindrical interpolation cutting point compensation

	#7	#6	#5	#4	#3	#2	#1	#0
19530		**CYS**	**CYA**					

[Input type] Parameter input
[Data type] Bit path

#5 CYA Specifies whether to perform cylindrical interpolation cutting point compensation in the cylindrical interpolation command (G07.1).
0: Perform.
1: Do not perform.

#6 CYS Specifies whether when the cylindrical interpolation cutting point compensation function is used, cutting point compensation is performed between blocks or together with a block movement if the cutting point compensation value is less than the setting of parameter No. 19534.
0: Performed between blocks.
1: Performed together with a block movement if the cutting point compensation value is less than the setting of parameter No. 19534.

19531	**Tool offset axis number for the XY plane**
19532	**Tool offset axis number for the ZX plane**
19533	**Tool offset axis number for the YZ plane**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes
Specify a tool offset axis that intersects the cylindrical rotation axis at right angles.

19534	**Limit for changing cylindrical interpolation cutting point compensation in a single block**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 1 to 999999999
The following operation is performed, depending on the setting of parameter No. 19530:
(1) Bit 6 (CYS) of parameter No. 19530) is set to 0

If the amount of cylindrical interpolation cutting point compensation is smaller than the value set in this parameter, cylindrical interpolation cutting point compensation is not performed. Instead, this ignored amount of cylindrical interpolation cutting point compensation is added to the next amount of cylindrical interpolation cutting point compensation to determine whether to perform cylindrical interpolation cutting point compensation.

(2) Bit 6 (CYS) of parameter No. 19530) is set to 1
If the amount of cylindrical interpolation cutting point compensation is smaller than the value set in this parameter, cylindrical interpolation cutting point compensation is performed together with the movement of the specified block.

> **NOTE**
> Set this parameter as follows:
> Setting < (setting for a rotation axis in parameter No. 1430) × 4/3
> where 4/3 is a constant for internal processing.

19535	**Limit of travel distance moved with the cylindrical interpolation cutting point compensation in the previous block unchanged.**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 1 to 999999999

The following operation is performed, depending on the type of interpolation:

(1) For linear interpolation
If the travel distance in a specified block is smaller than the value set in this parameter, machining is performed without changing the cylindrical interpolation cutting point compensation in the previous block.

(2) For circular interpolation
If the diameter of a specified arc is smaller than the value set in this parameter, machining is performed without changing the cylindrical interpolation cutting point compensation in the previous block. Cylindrical interpolation cutting point compensation is not performed according to a circular movement.

Alarm and message

Number	Message	Description
PS0015	TOO MANY SIMULTANEOUS AXES	A move command was specified for more axes than can be controlled by simultaneous axis control. Divide the number of programmed move axes into two blocks.
PS0175	ILLEGAL G07.1 AXIS	An axis which cannot perform cylindrical interpolation was specified. More than one axis was specified in a G07.1 block. An attempt was made to cancel cylindrical interpolation for an axis that was not in the cylindrical interpolation mode. For the cylindrical interpolation axis, set not 0 but one of 5, 6 or 7 (parallel axis specification) to parameter No. 1022 to instruct the arc with axis of rotation (bit 0 (ROT) of parameter No. 1006 is set to 1 and parameter No. 1260 is set) ON.
PS0176	ILLEGAL G-CODE USE(G07.1 MODE)	A G code was specified that cannot be specified in the cylindrical interpolation mode. This alarm also is generated when an 01 group G code was in the G00 mode or code G00 was instructed. Cancel the cylindrical interpolation mode before instructing code G00.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Cylindrical interpolation

6.9 POLYGON TURNING

T

Polygon turning means machining a workpiece to a polygonal figure by rotating the workpiece and tool at a certain ratio.

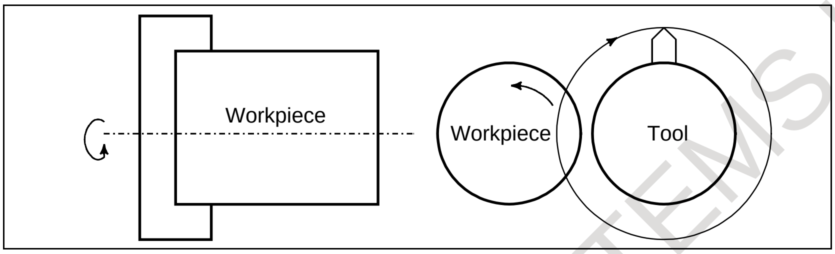


Fig. 6.9 (a)

By changing conditions which are rotation ratio of workpiece and tool and number of cutters, the workpiece can be machined to a square or hexagon. The machining time can be reduced as compared with polygonal figure machining using the polar coordinate interpolation.
The machined figure, however, is not exactly polygonal. Generally, polygon turning is used for the heads of square and/or hexagon bolts or hexagon nuts.
As the tool rotary axis, one of the following can be used:

- CNC controlled axis (servo axis)
- Second spindle (Two serial spindles are connected.)

Polygon turning performed using a servo axis as the tool rotary axis is referred to as polygon turning. Polygon turning performed using the second spindle as the tool rotary axis is referred to as polygon turning with two spindles.

Table 6.9 (a)

Function name	Workpiece axis	Tool rotary axis
Polygon turning	Spindle (Either an analog spindle or serial spindle is usable. However, a detector equivalent to a position coder is required.)	Servo axis
Polygon turning with two spindles	Spindle (Serial spindle)	Spindle (Serial spindle)

The specifications of polygon turning are described below.
For details of polygon turning with two spindles, see the specifications of polygon turning with two spindles.

6.9.1 Polygon Turning

Overview

A CNC controlled axis (servo axis) is assigned to the tool rotary axis.
This rotary axis of tool is called Y-axis in the following description. As the workpiece axis (spindle), either a serial spindle or analog spindle can be used.
The Y-axis is controlled by the G51.2 command, so that the ratio of the rotation speeds of the spindle (previously specified by S-command) and the tool becomes the specified ratio.
When simultaneous start is specified by G51.2, the one-rotation signal sent from the position codes set on the spindle is detected. After one-rotation signal detection, the Y-axis is controlled using the rotation ratio

of the spindle and Y-axis specified by P and Q. So, a position coder needs to be attached to the spindle. This control will be maintained until the polygon turning cancel command is executed (G50.2). Polygon turning is cancelled by any of the following in addition to the G50.2 command:

(1) Power off
(2) Emergency stop
(3) Servo alarm
(4) Reset (external reset signal ERS <Gn008.7>, reset/rewind signal RRW <Gn008.6>, and  key on the MDI panel)
(5) Occurrence of alarm PS0217 to PS0221, PS0314, and PS5018

NOTE

1 Before polygon turning, reference position return operation on the Y-axis needs to be specified to determine the rotation start position of the tool. This reference position return operation is performed by detecting a deceleration limit as in the case of manual reference position return operation. (By setting bit 7 (PLZ) of parameter No. 7600, reference position return operation can be performed without detecting a deceleration limit.)
2 The rotation direction on the Y-axis is determined by the sign of Q, and is not affected by the rotation direction of the position coder.
3 Among Y-axis position indications, the indication of a machine coordinate (MACHINE) changes within the amount of movement for 0 to 1 revolution as a movement is made on the Y-axis. The absolute coordinate and relative coordinate are not updated. So, when specifying an absolute-position command for the Y-axis after polygon turning mode cancellation, set a workpiece coordinate system after reference position return operation.
4 For the Y-axis engaged in polygon turning, jog feed and handle feed are disabled.
5 For the Y-axis not engaged in polygon turning, a move command can be specified as in the case of other controlled axes.
6 The Y-axis engaged in polygon turning is not counted in the number of simultaneously controlled axes.
7 One workpiece must be machined using a fixed spindle speed until the workpiece is finished.
8 G50.2 is the G code for suppressing buffering.
9 The following functions must be commanded with polygon turning cancel mode.
   - Inch/metric conversion
   - AI contour control
   - Tilted working plane indexing
   - Flexible path axis assignment

- Spindle connection

A position coder must be mounted on the spindle. However, polygon turning requires no additional changes to the spindle connection.

Polygon turning uses the position coder feedback signal to control the positional relationship (cutting position) between the spindle and tool rotation axis, and the ratio of speed.

- Tool rotation axis (servo axis) connection

Parameter No. 7610 specifies the controlled axis (servo axis) to be used as the tool rotation axis.

The same parameter setting as for ordinary servo axes applies to the servo axis connection for polygon turning except for some parameters.

When the machine is not in the polygon turning mode, the servo axis specified as the rotation tool axis functions as a feed axis. So, the servo axis can be:

- Used as a subspindle under PMC axis control
- Positioned by a move command from a machining program.

However, be careful about the angle to rotate through and feedrate.
Refer to the concrete example of the following a parameter setting.

- Principle of polygon turning

The principle of polygon turning is explained below. In the Fig. 6.9.1 (a), the radius of tool and workpiece are A and B, and the angular speeds of tool and workpiece are α and β. The origin of XY Cartesian coordinates is assumed to be the center of the workpiece.
Simplifying the explanation, consider that the tool center exists at the position Po (A, 0) on the workpiece periphery, and the tool nose starts from position Pto (A-B, 0).

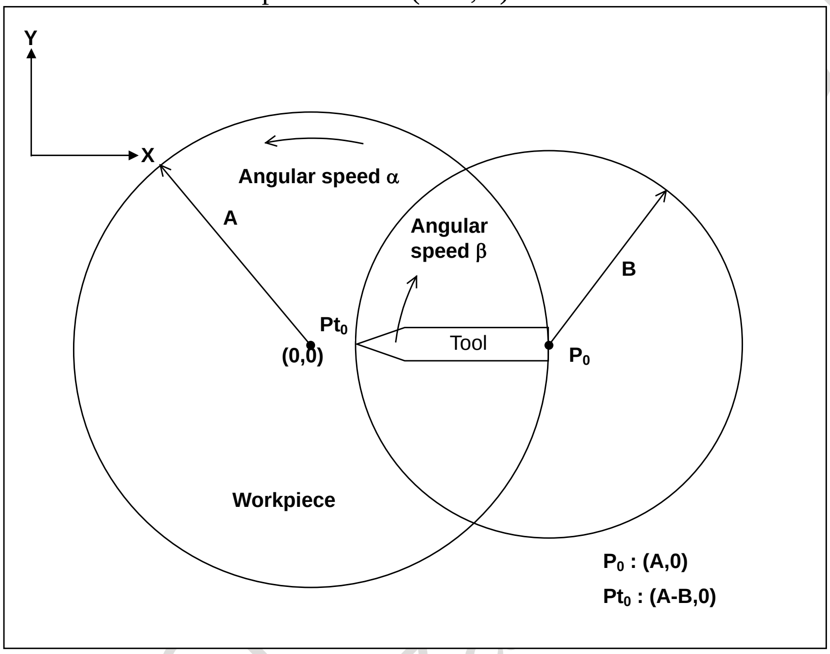


Fig. 6.9.1 (a)

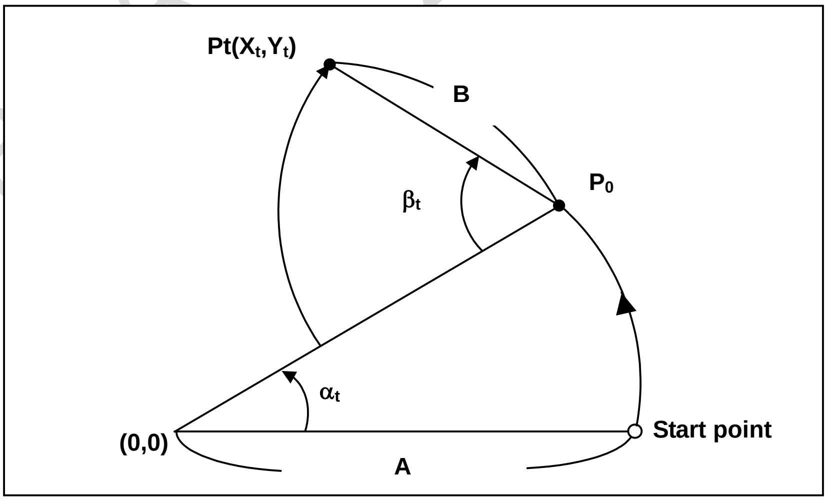


Fig. 6.9.1 (b)

In this case, the tool nose position Pt (Xt, Yt) after time t is expressed by equations 1 and 2:

$Xt=A\cos\alpha t-B\cos(\beta-\alpha)t$ (Equation 1)

$Yt=A\sin\alpha t+B\sin(\beta-\alpha)t$ (Equation 2)

Assuming that the rotation ration of workpiece to tool is 1:2, namely, β=2α, equations 1 and 2 are modified as follows:

$Xt=A\cos\alpha t-B\cos\alpha t=(A-B)\cos\alpha t$ (Equation 1)'

$Yt=A\sin\alpha t+B\sin\alpha t=(A+B)\sin\alpha t$ (Equation 2)'

These equations indicate that the tool nose path draws an ellipse with longer diameter A+B and shorter diameter A-B.

Then consider the case when one tool is set at 180° symmetrical positions, for a total of two. A square can be machined with these tools as shown Fig. 6.9.1 (c).

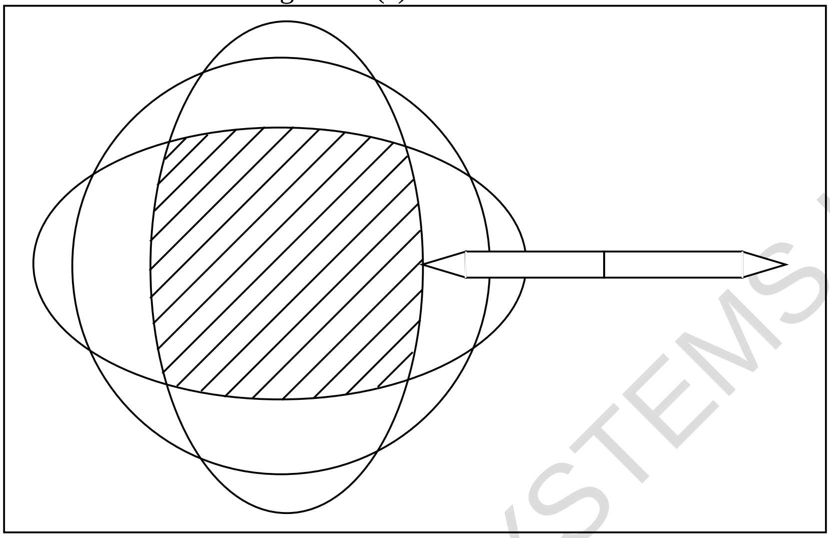


Fig. 6.9.1 (c)

If three tools are set at every 120°, the machining figure will be a hexagon as shown Fig. 6.9.1 (d).

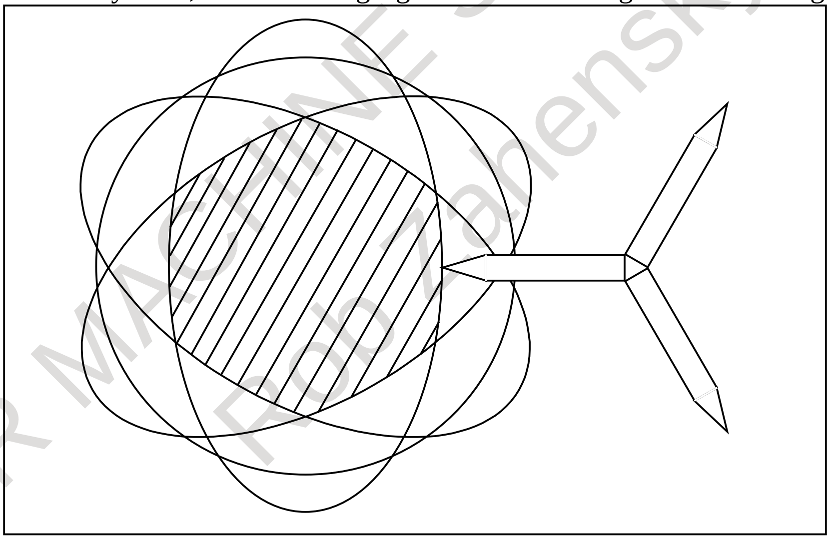


Fig. 6.9.1 (d)

> ⚠ **WARNING**
> For the maximum rotation speed of the tool, see the instruction manual supplied with the machine. Do not specify a spindle speed higher than the maximum tool speed or a ratio to the spindle speed that results in a speed higher than the maximum tool speed.

- Examples of parameter setting

The following descriptions exemplify typical parameter setting for polygon turning using a serial pulse coder (with a million pulse capability).

(1) Tool rotation axis setting

This example uses the CNC's fourth axis (connected as the Y-axis) as a rotation tool axis for polygon turning.

Parameter No. 7610 = 4 (controlled axis number for the tool rotation axis)

The following description assumes that the axis type parameter is set to the fourth axis.

(2) Servo parameter setting

Set the servo parameters as listed below:
CMR=1
DMR=36/100
(With the above setting, the reference counter capacity is 360000.)

Parameter No.1820=2	(CMR)
Parameter No.1821=360000	(reference counter capacity)
Parameter No.2084=36	(DMR numerator)
Parameter No.2085=100	(DMR denominator)

For the other servo parameters, specify typical values.

(3) Parameter setting for polygon turning

The least command increment, detection unit, the angle to rotate through per rotation for the Y axis are as follows:

Least command increment= $\dfrac{L \times \mathrm{CMR}}{\mathrm{Q} \times \mathrm{DMR}}$

Detection unit= $\dfrac{\text{Least command increment}}{\mathrm{CMR}} = \dfrac{\mathrm{L}}{\mathrm{Q} \times \mathrm{DMR}}$

Angle to rotate through per tool axis rotation=

$\dfrac{360}{\text{Least command increment}}$

where

L: Tool axis rotation angle per motor rotation (degrees),
(360 × speed increment ratio)

When the servo motor is connected directly to the rotation tool, for example, L = 360. When the tool speed is doubled, L = 720.

Q: Number of pulses per pulse coder rotation
(For a serial pulse coder, Q = 1000000.)

The least command increment specified here is specific to the Y axis. It is determined regardless of what is specified in parameter No. 1013#0,#1 (ISA/ISC). However, both ISA and ISC must be set to 0 for IS-B setting.

If the servo motor is connected directly to the rotation tool:

Least command increment= $\dfrac{360 \times 1}{1000000 \times 36/100} = 0.001$ [deg]

Detection unit=0.001 [deg]

Angle to rotate through per tool axis rotation= $\dfrac{360}{0.001} = 360000$

The upper limit to the tool rotation axis speed is:
Maximum servo motor speed × speed increment ratio

Therefore, if the maximum servo motor speed is 1000 [min^{-1}], and the servo motor is directly connected to the servo motor:

Upper limit to the tool rotation axis speed = 1000 × 1 = 1000 [min^{-1}]

This means the parameters must be set as follows:
Parameter No. 7620 = 360000 (angle to rotate through per tool axis rotation)
Parameter No. 7621 = 1000 (upper limit to tool rotation axis speed)

(4) Feedrate parameter setting

Because the least command increment is 0.01 [deg], the input unit for the feedrate is 1 [deg/min].
To obtain a rapid traverse speed of 1000 [min^{-1}], for example, specify as follows:
Parameter No. 1420 = 360000 (= 1000 × 360)
Other feedrate parameters must be set in degrees per minute.

(5) Commands from the NC program

When the machine is not performing polygon turning, the machining program can issue move commands to the Y-axis.

Such commands can be issued in the same way as for ordinary axes. However, note the travel increment and feedrate.
The Y-axis rotates through 0.003[deg] by the following command:
V3;
Likewise, the polygon axis rotates through 1.000[deg] by the following command:
V1.0;
The current position of the Y-axis in the machine coordinate system is normalized according to the value specified by parameter No. 7620.
Typical values range from 0.000 to 359.999.

Signal

Polygon synchronization under way signal PSYN<Fn063.7>

[Classification] Output signal
[Function] Informs the PMC that the machine is in the polygon turning mode.
[Output cond.] The polygon synchronization signal is set to logical “1” by the polygon turning mode command (G51.2) and stays at 1 during the polygon turning mode.
The signal is reset to logical “0” by the polygon turning mode reset command (G50.2) or a reset. It stays at logical 0 when the machine is not in the polygon turning mode.

> **NOTE**
> This signal uses the same address for both polygon turning (using the servo axis) and polygon turning with two spindles.

Other signals

Some signals related to the CNC controlled axis used as the tool rotation axis may be made ineffective depending on whether the machine is in the polygon turning mode.

Signal	Valid / invalid for the Y-axis of polygon turning mode
All-axis machine lock signal MLK<Gn044.1> Each-axis machine lock signal MLK1~MLK8 <Gn108>	Valid
Servo off signal SVF1~SVF8<Gn126>	Valid
Automatic operation stop signal *SP<Gn008.5>	Invalid
Interlock signal for all axes *IT <Gn008.0> Interlock signal for each axis *IT1~*IT8 <Gn130> Interlock signal for each axis direction +MIT1~+MIT8<Gn132>, -MIT1~-MIT8<Gn134>	Invalid
Feedrate override signal *FV0~*FV7 <Gn012>	Invalid
Dry run signal DRN<Gn046.7>	Invalid

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn063	PSYN							

Diagnosis data

478	Speed of the tool rotating axis during polygon turning (1/min)

This indication is the speed of the tool rotating axis during polygon turning.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7600	PLZ							PFF

[Input type] Parameter input

[Data type] Bit path

#0 PFF In spindle-servo polygon turning, feed forward for the tool rotation axis (servo axis) during polygon turning is always:
0: Disabled.
1: Enabled.

#7 PLZ Reference position return based on a G28 command on the tool rotation axis for polygon turning is:
0: Performed in the same sequence as manual reference position return.
1: Performed by positioning using the rapid traverse rate.
The synchronous axis returns to the reference position in the same sequence as the manual reference position return when no return-to-reference position is performed after the power is turned on.

	#7	#6	#5	#4	#3	#2	#1	#0
7603					PLROT			RPL

[Input type] Parameter input
[Data type] Bit path

#0 RPL Upon reset, polygon turning mode is:
0: Released.
1: Not released.

> **NOTE**
> 1 When an emergency stop occurs, the polygon turning mode is released regardless of whether this parameter is set to 0 or 1.
> 2 When any of the following PS alarms is issued, the polygon turning mode is released regardless of whether this parameter is set to 0 or 1:
> - Alarm PS0217, "DUPLICATE G51.2(COMMANDS)"
> - Alarm PS0219, "COMMAND G51.2/G50.2 INDEPENDENTLY"
> - Alarm PS0220, "ILLEGAL COMMAND IN SYNCHR-MODE"
> - Alarm PS0221, "ILLEGAL COMMAND IN SYNCHR-MODE"
> - Alarm PS5018, "POLYGON SPINDLE SPEED ERROR"
> 3 If an SV alarm is issued, the polygon turning mode is released regardless of whether this parameter is set to 0 or 1.
> 4 When this parameter is set to 1, polygon turning modal information is kept regardless of whether bit 6 (CLR) of parameter No. 3402 is set to 0 or 1.
> 5 Set bit 4 (C20) of parameter No. 3408 to 0.

#3 PLROT The machine coordinates of a tool rotation axis for polygon turning are:
0: Rounded by the setting in parameter No.7620.
1: Rounded by 360° (or the setting in parameter No. 1260 when bit 0 (ROA) of parameter No. 1008 is set to 1).

7610	Control axis number of tool rotation axis for polygon turning

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes

This parameter sets the control axis number of a rotation tool axis used for polygon turning.
However, when a G51.2 command is executed by setting 0 in this parameter, operation stops with the alarm PS0314.

7620	Movement of tool rotation axis per revolution for polygon turning

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the movement of a tool rotation axis per revolution.

7621	Maximum allowable speed for the tool rotation axis for polygon turning

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min^{-1}
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the maximum allowable speed of the tool rotation axis.

NOTE
If the speed of the tool rotation axis exceeds the set maximum allowable speed during polygon turning, the synchronization between the spindle and tool rotation axis is lost, and operation stops with alarm PS5018.

Alarm and message

Number	Message	Description
PS0217	DUPLICATE G51.2(COMMANDS)	G51.2 is further commanded in the G51.2 mode. Modify the program.
PS0218	NOT FOUND P/Q COMMAND	P or Q is not commanded in the G51.2 block, or the command value is out of the range. Modify the program. For a polygon turning between spindles, more information as to why this alarm occurred is indicated in diagnosis data No. 471.

Number	Message	Description
PS0219	COMMAND G51.2/G50.2 INDEPENDENTLY	G51.2 and 50.2 were specified in the same block for other commands. Modify the program in another block.
PS0220	ILLEGAL COMMAND IN SYNCHR-MODE	In polygon turning, movement is commanded by the NC program or PMC axis control interface for the polygon axis. Modify the program or check the PMC ladder.
PS0221	ILLEGAL COMMAND IN SYNCHR-MODE	Polygon turning and Cs contour control or balance cutting are executed at a time. Modify the program.
PS0314	ILLEGAL SETTING OF POLYGONAL AXIS	An axis was specified invalidly in polygon turning. For polygon turning: A tool rotation axis is not specified. (Parameter No. 7610) For polygon turning with two spindles: Valid spindles are not specified. (Parameter Nos. 7640 to 7643) - A spindle other tan the serial spindle. - A spindle is not connected. For concurrent use of polygon turning and polygon turning with two spindles: - In the polygon turning mode, the value of parameter No. 7605 (selecting the type of polygon turning) was changed. - An attempt is made to use a spindle used for polygon turning also for polygon turning with two spindles.
PS5018	POLYGON SPINDLE SPEED ERROR	In G51.2 mode, the speed of the spindle or polygon synchronous axis either exceeds the clamp value or is too small. The specified rotation speed ratio thus cannot be maintained. For polygon turning with two spindles: More information as to why this alarm occurred is indicated in diagnosis data No. 471.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Polygon turning

6.9.2 Polygon Turning with Two Spindles

Overview

When two or more serial spindles are used, the workpiece rotation axis (master axis) and tool rotation axis (polygon synchronization axis) are synchronized at a certain speed ratio.
With this function, it is also possible to specify the phase difference between the master and polygon synchronization axes.

The polygon turning with two spindles can use different spindle speeds for the same workpiece, because it performs automatic phase compensation when a polygon synchronization mode command is issued or the S command is changed during polygon synchronization mode.

With a multiple path system, polygon turning is possible on each path.

By default, the first and second spindles in each system are selected as the master axis and polygon synchronization axis for each system. By setting parameters No. 7640 and No. 7641, however, any spindles belonging to the same system can be selected as the master axis and polygon synchronization axis. By setting parameters No. 7642 and No. 7643, however, any spindles belonging to different systems can be selected as the master axis and polygon synchronization axis.

- Command format

The following shows the program command format for polygon turning with two spindles. This is the same as the program command format for polygon turning except for the following points.

1) The command position (R) can be used.
2) Repeated specification in polygon synchronization mode is allowed.

For polygon turning, see also the section about polygon turning.

- Mode command and command value change

The G51.2 command is modal. Once specified, the P, Q, and R values stay unchanged until another G51.2 is issued to change them or polygon synchronized mode is released.

In polygon synchronized mode, the S command for the master axis controls the polygon synchronization axis so that it rotates at a speed of S × Q/P and its phase is R.

- Release command

G50.2 command releases the polygon synchronization mode. This mode is released also when:

<1> Reset and emergency stop
(Setting bit 0 (RPL) of parameter No.7603 to 1 prevents polygon synchronization mode from being released.)
<2> Power is turning off.
<3> An alarm condition occurs in the spindle control unit, and the serial spindle control unit stops in an emergency on the PMC signals *ESPA<Gn071.1> and *ESPB<Gn075.1>.
<4> Alarm PS0218, PS0219, PS0221, PS0314, or PS5018 occurs

NOTE
1 G51.2 and G50.2 must be issued separately from other commands.
2 In a G51.2 issued to enter the polygon synchronization mode, R is omissible, but P and Q are required.
3 When G51.2 is specified to change the P, Q, and R settings during the polygon synchronization mode, R can be specified independently. When P and Q are changed, however, be sure to specify both P and Q.
4 When bit 0 (RPL) of parameter No.7603 = 0 regardless of the setting of bit 6 (CLR) of parameter No.3402, a reset issues G50.2 (polygon synchronization release mode). When bit 0 (RPL) of parameter No. 7603 = 1, a reset does not clear G51.2 (polygon synchronization mode) in polygon synchronization mode.
5 The following functions must be commanded with polygon turning cancel mode.
- Inch/metric conversion
- AI contour control
- Tilted working plane indexing
- Flexible path axis assignment

- Spindle operation during the polygon synchronization mode with two spindles

When the start of polygon synchronization is specified by G51.2, the polygon synchronization axis speed is adjusted to Q/P times the master axis speed based on spindle speed command value for the master axis and then phase matching is performed.

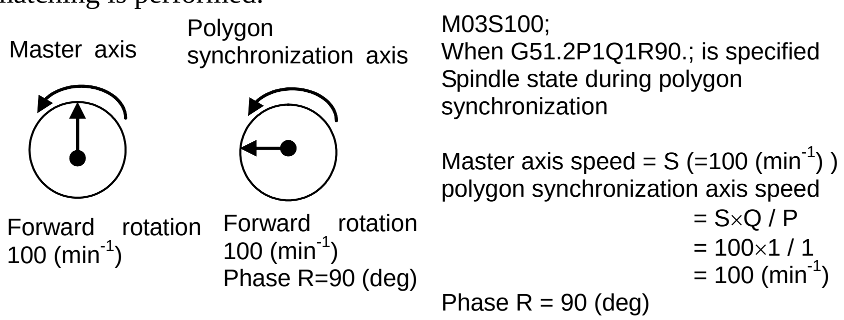


Fig. 6.9.2 (a)

When the spindle is accelerated or decelerated or when phase matching is being performed, the synchronization state of rotation ratio of P to Q is not guaranteed. Therefore, in an actual cutting block command, control the spindle speed arrival signal SAR<Gn029.4> after checking that the signal state of the polygon spindle speed arrival signal for polygon turning with two spindles PSAR<Fn063.2> is 1 or take sufficient wait time in the program.

Each time any of the following changes is made during polygon synchronization, control the speed and then perform phase matching.

1) The spindle speed command for the master axis is changed.
2) The speed ratio commands P and Q are specified by G51.2 again.
3) The phase command value R is specified by G51.2 again.
4) The polygon spindle stop signal *PLSST <Gn038.0> is changed from “0” to “1”.

How to specify the spindle speed for the master axis (multi-spindle control or spindle output control by PMC) does not change regardless of whether polygon synchronization mode or normal operation is selected. Spindle speed specification for the polygon synchronization axis is disabled in polygon synchronization mode.

If the speed of the polygon synchronization axis (S × Q / P(min^{-1}): where S is the master axis speed) exceeds the clamp speed set in parameter No. 7621 after the command is executed, an alarm (PS5018) occurs and the polygon synchronization state is cleared.

The diagnostic display bit 2 (QCL) of No. 471 can be used to check whether the clamp speed is exceeded after the command is executed.

- PMC sequence

Although this function is based on the G-code system, it is necessary to add or change PMC ladder sequences because control on the part of the spindle is also required.
(See the description of the PMC sequence in this section.)

- Example of polygon turning with two spindles

This example of polygon turning with two spindles produces a square using single-edged cutting tools (for roughing and finishing).

.		
Mxy; Txxyy;	Step 1.	Mount a roughing tool on the polygon synchronization axis (tool rotation axis).
G00 X100. Z20. M03 S1000 ;	Step 2.	Start rotating the workpiece (with the master axis at 1000 (min^{-1}))
.		
G51.2 P1 Q2 R0 ;	Step 3.	Start rotating the tool. In the PMC ladder, the polygon synchronization axis is energized by polygon synchronization under way signal PSYC<Fn063.7>. After acceleration to 2000 (min^{-1}) is performed as the polygon synchronization axis, positioning is performed. The phase is 0. In the PMC ladder, the spindle speed arrival signal SAR<Gn029.4> is controlled based on the polygon spindle speed arrival signal PSAR<Fn063.2> during the polygon synchronization mode.
G01 X80. F10. ;	Step 4.	Starts cutting along the X-axis after SAR<Gn029.4> becomes logical 1 in signal control at step 3.
.		
G04 P4000 ;	Step 5.	Polygonal turning (roughing 1)
.		
G00 X100. ;	Step 6.	Retract the tool along the X-axis.
.		
G51.2 R180.;	Step 7.	Change the phase by 180 degrees.
.		
[Repeat steps 4, 5, and 6.]	Step 8.	Polygonal turning (roughing 2)
.		
G50.2 ; Myz; Tyyzz;	Step 9.	Release the polygon synchronization mode. Change to a finishing tool.
.		
S2000; G51.2 P1 Q2 R0 ; . .	Step 10.	Change the spindle speed for finishing Start rotating the tool. (master axis at 2000 min^{-1} and polygon synchronization axis at 4000 min^{-1} with a phase difference of 0).
[Repeat steps 4, 5, and 6.]	Step 11.	Polygonal turning (finishing 1)
.		
[Repeat step 7.]	Step 12.	Change the phase by 180 degrees.
.		
[Repeat steps 4, 5, and 6.]	Step 13.	Polygonal turning (finishing 2)
.		
G50.2 ; . . .	Step 14.	Release the polygon synchronization mode. The polygon synchronization axis (tool rotation axis) stops. The master axis rotates at 2000 (min^{-1}).
M05S0; . .	Step 15.	The master axis stops.

Diagnosis data

For polygon turning with two spindles, the following information is displayed on the diagnosis screen.

	#7	#6	#5	#4	#3	#2	#1	#0
470	SC0	LGE		SCF	PSC	PEN	PSU	SPL

Polygon turning with two spindles
Indication of information about the polygon synchronization mode

SPL Polygon synchronization with two spindles under way
PSU Polygon synchronization mode with two spindles being activated

> **NOTE**
> If only PSU becomes 1, but no change occurs, and the program stops in a block containing a G51.2 command, the speed of an spindle does not reach the targeted polygon synchronization speed, for example, because bit 7 (PST) of parameter No. 7603 = 0 keeps the spindle from being energized.

PEN Polygon synchronization mode with two spindles released
PSC Spindle speed being changed during polygon synchronization mode with two spindles
SCF Spindle speed has been changed during polygon synchronization mode with two spindles
LGE The loop gain is different between the spindles during polygon synchronization mode with two spindles.

> **NOTE**
> When the speed is changed during polygon synchronization mode, LGE is set to 1 if the spindle synchronization control loop gain used by the serial spindle control unit is different between the master spindle and polygon synchronization axis.
> Diagnosis display indicates the loop gain because this function requires that both spindles be controlled with the same loop gain. However, no alarm is issued even if the loop gain is different between the spindles.
> (For the serial spindle control unit, the parameters used are changed according to the state of the CTH1 and CTH2 signals.)

SC0 Actual speed command is 0 during polygon synchronization mode with two spindles.

> **NOTE**
> Signal SC0 is not a value specified by the program. It is set to 1 under any of the following conditions:
> 1. When the S command value is adjusted according to the signals related to spindle control, SSTP<Gn029.6> and SOV0-SOV7<Gn030> and the signal related to multi-spindle control <Gn027>, the result is 0.
> 2. The S command value is smaller than the spindle control resolution (the result of multiplying the S command value by a value of 4095/(maximum spindle speed) is less than 1).
>   The S command value is specified by SIND control <Gn032, Gn033>, and it is 0.
>
> If SC0 = 1, the spindle speed becomes 0 and bit 0 (SCU) of diagnosis data No. 471 becomes 1. In this case, the polygon synchronization rotation ratio is impractical, but alarm PS5018 does not occurs, because it is regarded as the result of the command.

If the following status is indicated during the polygon synchronization mode, there are no abnormalities.

	#7	#6	#5	#4	#3	#2	#1	#0
470	0	0	0	1	-	0	0	1

	#7	#6	#5	#4	#3	#2	#1	#0
471	NPQ	PQE	QMS	NSP	SUO	QCL		SCU

Polygon turning with two spindles
Indication of causes for alarms PS5018, PS0314, and PS0218

#0 to #3 Causes for alarm PS5018
Alarm PS5018 is cleared by a reset, but the indication of its causes remains until the causes are cleared or the polygon synchronization mode is released.

SCU The specified speed is too low during polygon synchronization mode with two spindles.
(The unit of speed calculated internally becomes 0.)

NOTE
SCU becomes 1 also when the specified spindle speed is 0 (bit 7(SC0) of diagnosis data No.470 is 1). In this case, however, alarm PS5018 is not issued (because the command is 0). When bit 7(SC0) of diagnosis data No.470 is 0 and bit 0(SCU) of diagnosis data No.471 is 1, alarm PS5018 occurs. Normally this does not occur with speed at which the spindle can rotate.

QCL The polygon synchronization axis is clamped.

NOTE
QCL becomes 1, when the polygon synchronization axis receives a command with a polygon synchronization speed that is higher than the value specified in parameter No. 7621 and is clamped at that speed.

SUO The specified speed is too high during the polygon synchronization mode with two spindles.
(It is clamped to the upper limit calculated internally.)

NOTE
SUO occurs, if a result of (speed specified for the master spindle)/(value specified at P) is higher than 59998. In other words, the master spindle must rotate at a speed lower than 59998 min^{-1} assuming P = 1.

#4 Causes for alarm PS0314
When alarm PS0314 occurs, the polygon synchronization mode is released, but the indication of its causes remains until the alarm PS0314 is cleared by a reset.

NSP A spindle necessary for control is not connected.
(For example, there is not a serial spindle or the second spindle.)
The axis settings for polygon turning are not correct.

#5 to #7 Causes for alarm PS0218
When alarm PS0218 occurs, the polygon synchronization mode is released, but the indication of its causes remains until the alarm PS0218 is cleared by a reset.

QMS When bit 1 (QDR) of parameter No. 7603 = 1, a negative value is specified at Q.

PQE In a G51.2, either P or Q has a value out of the specifiable range.
Or, P and Q are not specified as a pair.

NPQ In a G51.2, R is specified when P and Q have not been specified at all, or none of P, Q, and R has been specified.

Indication of values specified during the polygon synchronization mode with two spindles

474	Rotation ratio for the master axis during the polygon synchronization mode with two spindles (P command value)

This indication is the current rotation ratio (P command value) of the master axis during the polygon synchronization mode with two spindles.

475	Rotation ratio for the polygon synchronization axis during the polygon synchronization mode with two spindles (Q command value)

This indication is the current rotation ratio (Q command value) of the polygon synchronization axis during the polygon synchronization mode with two spindles.

476	Phase difference between the two spindles under polygon synchronization control with two spindles(R command value)

This indication is the current phase value (R command value) specified during the polygon synchronization mode with two spindles. (The increment system for the reference axis used.)

However, if bit 5 (RDG) of parameter No. 7603 = 1, the indication is the amount of shifting specified for the serial spindle (number of pulses after conversion is performed assuming 360 degrees = 4096 pulses).
Indication of the actual speed of each spindle during the polygon synchronization mode with two spindles

477	Actual master axis speed (min^{-1}) during the polygon synchronization mode with two spindles

This indication is the actual speed of the master axis during the polygon synchronization mode with two spindles.

478	Actual polygon synchronization axis speed (min^{-1}) during the polygon synchronization mode with two spindles

This indication is the actual speed of the polygon synchronization axis during the polygon synchronization mode with two spindles.

Spindle tuning screen

During the polygon synchronization mode with two spindles, information for the spindle synchronization control mode is displayed.

The following information is displayed on the spindle tuning screen.
”Positional deviation” indicates the positional deviation of the spindle currently being displayed.
”Synchronous deviation” indicates the absolute value of a synchronization error in polygon synchronization during polygon turning with two spindles. The synchronization error value is displayed on the spindle tuning screen for the spindle used as the polygon synchronization axis.
For details, please refer to Spindle tuning screen of MAINTENANCE MANUAL (B-64695EN).

Motor speed
Spindle speed
Positional deviation S1
Positional deviation S2
Synchronous deviation

Signal

Polygon spindle stop signal *PLSST<Gn038.0>

[Classification] Input signal

[Function] This function is enabled when bit 7 (PST) of parameter No. 7603 is set to 1. This signal is used to stop the spindle during the polygon synchronization mode with two spindles.
0 = polygon spindle stop
1 = polygon spindle operable
During the polygon synchronous mode with two spindles, the rotation of the spindle is controlled with a position loop created.
The spindle is rotated by move command pulses based on the polygon synchronization speed.
When the activated spindle is turned off by the spindle stop (for example, M05) command, move command pulses are generated even when activation is off unless the spindle speed command is set to 0 using the spindle stop signal *SSTP<Gn029.6> or the like, some error pulses being accumulated. When activation is turned on at this time, the spindle may rotates at high speed because the accumulated error pulses are moved at a time.
This signal can be used to stop the delivery to the polygon spindle only during the polygon synchronous mode with two spindles. The signal must be controlled according to the activation status (on/off) of the polygon spindle.

For details on usage, see the section that describes PMC sequences.

Polygon synchronization under way signal PSYN<Fn063.7>

[Classification] Output signal

[Function] Informs the PMC that the system is in the polygon synchronization mode.

[Output cond.] The polygon synchronization mode command (G51.2) sets this signal to logical “1”. It stays at 1 as long as the system is in the polygon synchronization mode. It is turned to 0 when the polygon synchronization mode is cleared (G50.2 command or a reset). It stays at 0 when the system is not in the polygon synchronization mode.

> **NOTE**
> The same address is used for this signal in both polygon turning (using the servo axis) and the polygon turning with two spindles.

Polygon spindle speed arrival signal PSAR<Fn063.2>

[Classification] Output signal

[Function] Informs the PMC that the spindle has reached its constant-speed for polygon synchronization during polygon turning with two spindles.

[Output cond.] During polygon synchronization mode with two spindles, whether the constant-speed is reached for polygon synchronization is output as shown below:

0 = not reached (during phase change or acceleration/deceleration under way)

1 = reached

During the polygon synchronization mode, this signal becomes logical 1 when the speed of each spindle reaches the acceptable level specified in parameter No. 7631 and remains there for a period specified in parameter No. 7632.

If the speed of either spindle goes off the acceptable level, or a change is made to the S command, the signal returns to logical “0” and begins monitoring the above condition.

When this signal is “0”, the specified speed ratio and phase are not guaranteed for polygon turning. If the signal is confirmed before actual turning is started, however, the operation is more efficient than when a dwell command (like G04) is used to allow wait time.

Polygon master axis not arrival signal PSE1<Fn063.0>
Polygon synchronization axis not arrival signal PSE2<Fn063.1>

[Classification] Output signal

[Function] Informs the PMC whether the actual speed of each spindle has reached the specified speed during polygon turning mode with two spindles.

[Output cond.] During the polygon synchronization mode with two spindles, whether each spindle has reached the polygon synchronization speed is output as shown below:

0 = reached

1 = not reached (during phase change or acceleration/deceleration under way)

During the polygon synchronization mode with two spindles, this signal becomes logical 1 when the speed of master axis and polygon synchronization axis does not reach the acceptable level specified in parameter No. 7631.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn038								*PLSST

	#7	#6	#5	#4	#3	#2	#1	#0
Fn063	PSYN					PSAR	PSE2	PSE1

PMC sequence

The following shows the signal status time chart when the polygon synchronization mode is on and off.

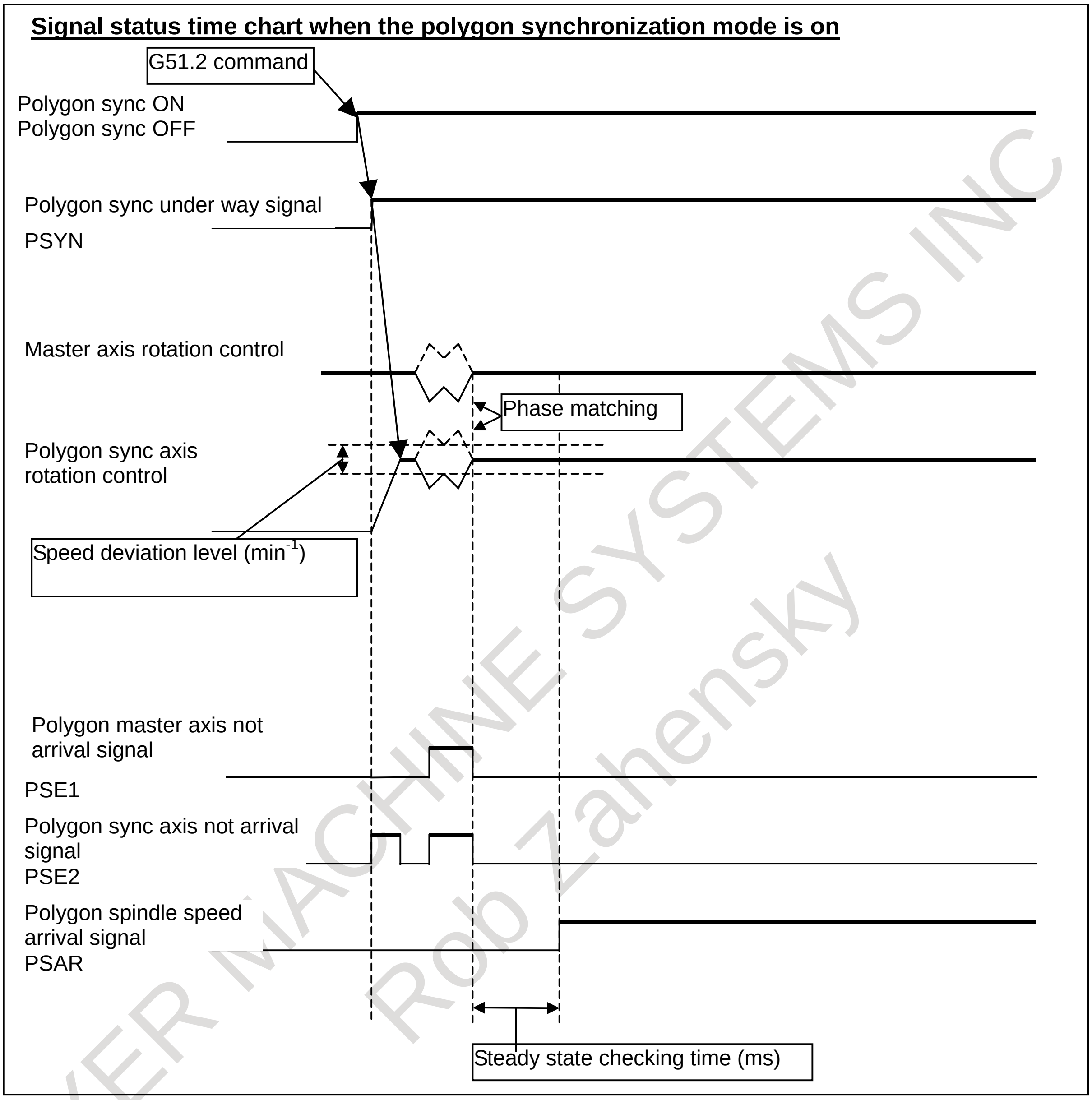


Fig. 6.9.2 (b)

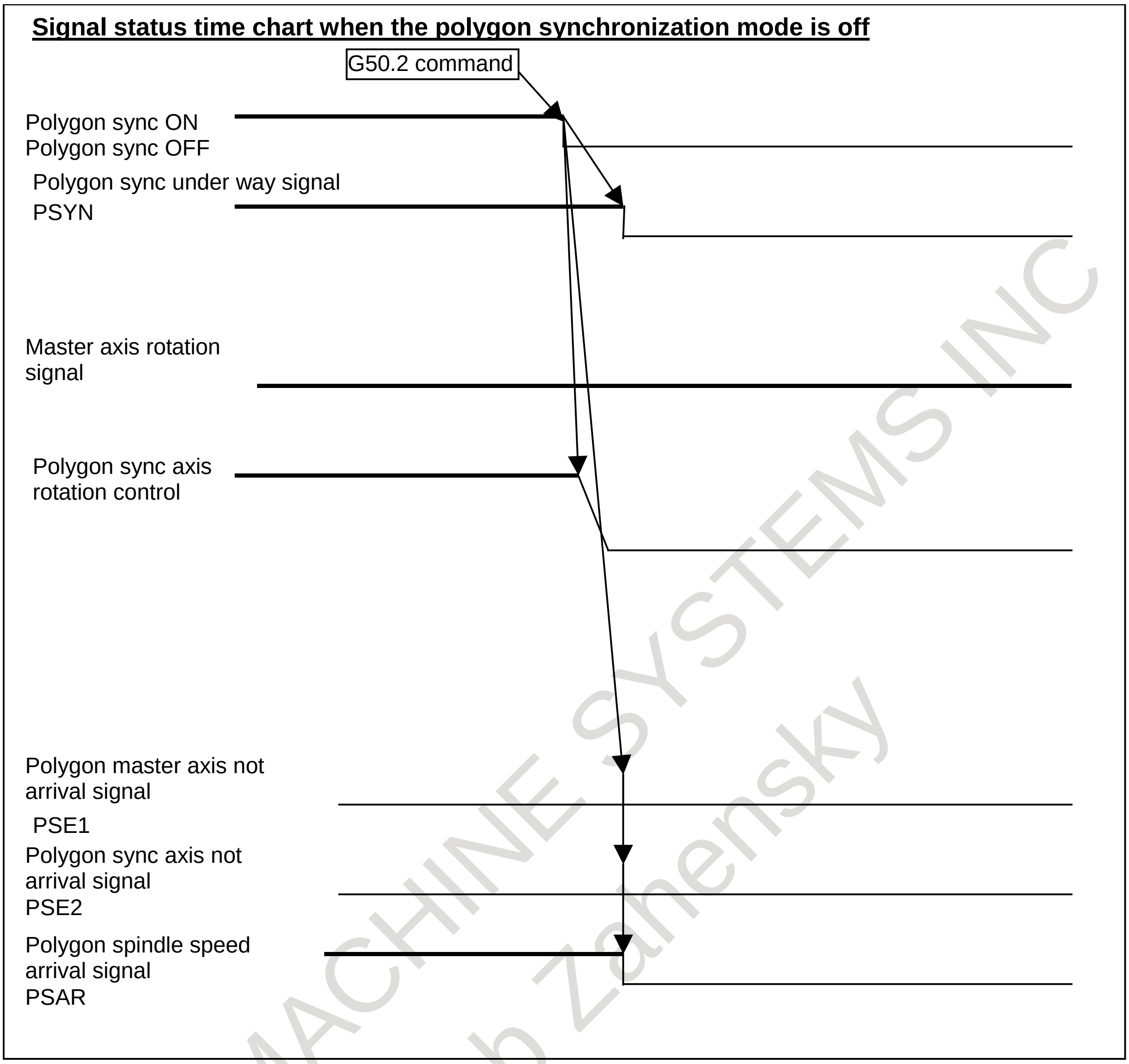


Fig. 6.9.2 (c)

When the polygon synchronization mode is started by the G51.2 command, the polygon synchronization under way signal PSYN<Fn063.7> is set to “1”.
Create a PMC sequence during the polygon synchronization mode by checking the polygon synchronization under way signal with the PMC ladder.

First, select one method of controlling spindle activation used during the polygon synchronization mode from methods A and B. Then create a PMC sequence according to the selected method.

- Method A

During the polygon synchronization mode, activation of the master axis and polygon synchronization axis is automatically turned on.
During this mode, activation of the spindle is not turned off.

For method A, set bit 7 (PST) of parameter No. 7603 to 0.
In a PMC sequence, activation of the master axis and polygon synchronization axis must be turned on when the polygon synchronization under way signal PSYN<Fn063.7> changes from “0” to “1”.

In addition, while the polygon synchronization under way signal is 1, prevent the activation of the master axis and polygon synchronization axis from being turned off by, for example, prohibiting the acceptance of spindle stop (for example M05) during normal control.

When the polygon synchronization under way signal PSYN<Fn063.7> changes from “1” to “0”, basically turn off activation.

The NC stops at the G51.2 command block for entering the polygon synchronization mode and does not proceed to the next block until the spindle speed reaches the polygon synchronization speed.

- Method B

Even during the polygon synchronization mode, another M code or the like is used to turn on or off activation of the spindle.
Or, even during the polygon synchronization mode, activation of the spindle may be turned off.

For method B, set bit 7 (PST) of parameter No. 7603 to 1.
This parameter setting enables the polygon spindle stop signal *PLSST<Gn038.0> and makes the NC proceed to the next block without waiting for the spindle speed to reach the polygon synchronization speed in the G51.2 command block for starting the polygon synchronization mode.

In a PMC sequence, during the polygon synchronization mode (the polygon synchronization under way signal PSYN<Fn063.7> is “1”), set the polygon spindle stop signal *PLSST<Gn038.0> to “1” after confirming that activation of the master axis and polygon synchronization axis is turned on.
When the polygon synchronization mode is not entered (the polygon synchronization under way signal PSYN<Fn063.7> is “0”), set the polygon spindle stop signal *PLSST<Gn038.0> to “0”.

When the polygon spindle stop signal *PLSST<Gn038.0> is changed from “0” to “1” during the polygon synchronization mode, phase control is performed after acceleration from the spindle stop state to the polygon synchronization speed is performed.

- Sequences common to methods A and B

During the polygon synchronization mode, observe the follow rules to create a PMC sequence regardless of whether method A or method B is used.

(1) The rotation direction of the master axis during the polygon synchronization mode is fixed to the forward direction rather than being switched by the forward/reverse rotation specification signal (SFR/SRV). When changing the rotation direction, use a command.
   - When using the S command in the program
     Specify M03/M04 by setting bit 7 (TCW) of parameter No. 3706 to 1.
   - When using a command by the spindle output control by the PMC
     Switch the rotation direction by setting the polarity selection signal SSIN<Gn033.6> to “1” to control the polarity specification signal SGN<Gn033.5>.
     The rotation direction of the polygon synchronization axis must also be fixed to the forward direction rather than being switched by the SFR/SRV signal.

(2) During the polygon synchronization mode, phase control is performed after the polygon synchronization speed is reached. Therefore, the speed arrival signal of each spindle (SARA<Fn045.3> and SARB<Fn049.3>) is insufficient for determining the start condition of the cutting feed command.
To confirm that the spindle speed is reached during the polygon synchronization mode, use the polygon spindle speed arrival signal PSAR<Fn063.2> to control the spindle speed arrival signal SAR<Gn029.4>.
The polygon spindle speed arrival signal PSAR<Fn063.2> can be used to check whether both spindles meet the settings of parameters No. 7631 and 7632 after phase control is performed in the polygon synchronization mode.

The set time (parameter No. 3740) that elapses before the spindle speed arrival signal is checked is also valid for the execution of the cutting feed command during the polygon synchronization mode.

If the spindle speed arrival signal SAR<Gn029.4> is not used as the start condition for the cutting feed command, the time required for the master axis and polygon synchronization axis to rotate at steady-state speed must be reserved during start of the polygon synchronization mode, change of the spindle speed, or change of the machining condition by a program command (for example, G04) before cutting by polygon turning is started.

(3) During the spindle polygon synchronization mode, the polygon synchronization axis cannot be rotated independently of the master axis.
Since the spindle orientation function (ORCMA <Gn070.6> and ORCMB<Gn074.6>) cannot also be used, the polygon synchronization mode must be released before shifting gears or changing tools or workpieces.
If necessary, take such an action that displays a message requesting the release of the polygon synchronization mode when an unavailable command is used during the polygon synchronization mode.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7603	PST		RDG				QDR	RPL

[Input type] Parameter input
[Data type] Bit path

#0 RPL Upon reset, polygon turning with two spindles mode is:
0: Released.
1: Not released.

#1 QDR The rotational direction of the polygon synchronization axis:
0: Depends on the sign (+/-) of a specified value for Q.
1: Depends on the rotational direction of the first spindle.
If a negative value is specified for Q when QDR = 1, the alarm PS0218 is issued.

#5 RDG On the diagnosis data No. 476, for spindle-spindle polygon phase command value (R), displays:
0: The specified value (in the increment system for the rotation axis).
1: The actual number of shift pulses.

> **NOTE**
> A phase command is specified in address R, in units of degrees. For control, the actual shift amount is converted to a number of pulses according to the conversion formula: 360 degrees = 4096 pulses. This parameter switches the display of a specified value to that of a converted value.

#7 PST The polygon spindle stop signal *PLSST <Gn038.0> is:
0: Not used.
1: Used.

7621	Maximum allowable speed for the tool rotation axis for polygon turning

[Input type] Parameter input
[Data type] 2-word path

[Unit of data] min^{-1}
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the maximum allowable speed of the tool rotation axis.

> **NOTE**
> If the speed of the tool rotation axis exceeds the set maximum allowable speed during polygon turning, the synchronization between the spindle and tool rotation axis is lost, and operation stops with alarm PS5018.

7631	Allowable spindle speed deviation level in polygon turning with two spindles

[Input type] Parameter input
[Data type] Word path
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999
This parameter sets the allowable level of deviation between the actual speed and specified speed of each spindle in polygon turning with two spindles. The value set with this parameter is used for both the master axis and polygon synchronization axis.
When 0 is set in this parameter, the specification of 8 [min^{-1}] is assumed.

7632	Steady state confirmation time duration in spindle polygon turning

[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 32767
This parameter sets the duration required to confirm that both spindles have reached their specified speeds in polygon turning with two spindles.
If the state where the speed of each spindle is within the range set with parameter No.7631, and has lasted at least for the duration specified with parameter No.7632, the spindle polygon speed arrival signal PSAR <Fn063.2> is set to "1".
When 0 is set in this parameter, the specification of 64 [msec] is assumed.

7640	Master axis in polygon turning with two spindles

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Within a path)
This parameter sets the master axis in polygon turning with two spindles.

> **NOTE**
> 1 Polygon turning with two spindles is enabled only for serial spindles.
> 2 When any one of parameters No. 7642 and 7643 is set to 0 and any one of parameters No. 7640 and 7641 is set to 0, polygon turning is performed using the first spindle (master axis) and the second spindle (polygon synchronous axis) in the path to which the parameter belongs.

> **NOTE**
> 3 When a spindle other than the first serial spindle is used as a master axis, the multi-spindle control is required to specify an S command for the master axis.
> 4 When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.
> 5 When this parameter is used, set the parameters No.7642 and No.7643 to 0.

7641	Polygon synchronous axis in polygon turning with two spindles

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Within a path)
This parameter sets the polygon synchronous (slave) axis in polygon turning with two spindles.

> **NOTE**
> 1 Polygon turning with two spindles is enabled only for serial spindles.
> 2 When any one of parameters No. 7642 and No. 7643 is set to 0 and any one of parameter No. 7640 and No. 7641 is set to 0, polygon turning is performed using the first spindle (master axis) and the second spindle (polygon synchronous axis) in the path to which the parameter belongs.
> 3 When a spindle other than the first serial spindle is used as a master axis, the multi-spindle control is required to specify an S command for the master axis.
> 4 When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.
> 5 When this parameter is used, set the parameters No.7642 and No.7643 to 0.

7642	Master axis in polygon turning with two spindles (spindle number common to the system)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Common to the system)
This parameter sets the master axis in polygon turning with two spindles.

NOTE

1 Polygon turning with two spindles is enabled only for serial spindles.
2 This parameter is invalid if either parameter No. 7642 or No.7643 is set to 0. In this case, the settings of parameter No. 7640 and No.7641 are valid.
3 When a spindle other than the first serial spindle is used as a master axis, the multi-spindle control is required to specify an S command for the master axis.
4 When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.
5 A spindle number common to the system is to be set in this parameter. When using this parameter, set 0 in parameter No. 7640 and No. 7641.

7643	Polygon synchronous axis in polygon turning with two spindles (spindle number common to the system)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Common to the system)

This parameter sets the polygon synchronous (slave) axis in polygon turning with two spindles.

NOTE

1 Polygon turning with two spindles is enabled only for serial spindles.
2 This parameter is invalid if either parameter No. 7642 or No.7643 is set to 0. In this case, the settings of parameter No. 7640 and No.7641 are valid.
3 When a spindle other than the first serial spindle is used as a master axis, the multi-spindle control is required to specify an S command for the master axis.
4 When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.
5 A spindle number common to the system is to be set in this parameter. When using this parameter, set 0 in parameter No. 7640 and No. 7641.

4032	Acceleration at spindle synchronous control

[Input type] Parameter input
[Data type] Word spindle
[Data unit] 1min^{-1}/sec
[Valid data range] 0 to 32767

This parameter sets an acceleration value for linear acceleration/deceleration when the synchronous speed command for spindle synchronous control is changed.
For polygon machining with two spindles, when bit 1 (POLYM) of parameter No. 4001 is 1, the allowable data range is not limited.

	Data range		
Spindle synchronous control	0-23767		
Polygon turning with two spindles	POLYM=0		POLYM=1
	SPDUNT=0	SPDUNT=1	0-32767
	0, 916-32767	0, 92-32767	

NOTE
1. Set exactly the same data for a master axis spindle and a slave axis. When different data is set, synchronization between the two spindles is not guaranteed.
2. When this parameter is set to 0, acceleration/deceleration of a spindle is not applied. Set an appropriate value.

4065	Position gain in servo mode (HIGH)
4066	Position gain in servo mode (MEDIUM HIGH)
4067	Position gain in servo mode (MEDIUM LOW)
4068	Position gain in servo mode (LOW)

[Data type] Word spindle
[Unit of data] 0.01 sec^{-1}
[Valid data range] 0 to 32767
These parameters set a servo loop gain on servo mode. (spindle positioning, rigid tapping, etc.)

NOTE
When the spindle positioning by a serial spindle is performed, set the position control loop gain in place of parameter No. 4970. For which gear is used, it depends on the clutch/gear signal (serial spindle) CTH1s, CTH1s.

Alarm and message

Number	Message	Description
PS0194	SPINDLE COMMAND IN SYNCHRO-MODE	A Cs contour control mode, spindle positioning command, or rigid tapping mode was specified during the spindle synchronous control mode or spindle command synchronous control mode.
PS0218	NOT FOUND P/Q COMMAND	P or Q is not commanded in the G51.2 block, or the command value is out of the range. Modify the program. For a polygon turning between spindles, more information as to why this alarm occurred is indicated in diagnosis data No. 471.
PS0219	COMMAND G51.2/G50.2 INDEPENDENTLY	G51.2 and 50.2 were specified in the same block for other commands. Modify the program in another block.
PS0221	ILLEGAL COMMAND IN SYNCHR-MODE	Polygon turning synchronous operation and Cs contour control or balance cutting are executed at a time. Modify the program.

Number	Message	Description
PS0314	ILLEGAL SETTING OF POLYGONAL AXIS	An axis was specified invalidly in polygon turning. For polygon turning: A tool rotation axis is not specified. (Parameter No. 7610) For polygon turning with two spindles: Valid spindles are not specified. (Parameter Nos. 7640 to 7643) - A spindle other than the serial spindle. - A spindle is not connected. For concurrent use of polygon turning and polygon turning with two spindles: - In the polygon turning mode, the value of parameter No. 7605 (selecting the type of polygon turning) was changed. - An attempt is made to use a spindle used for polygon turning also for polygon turning with two spindles.
PS5018	POLYGON SPINDLE SPEED ERROR	In G51.2 mode, the speed of the spindle or polygon synchronous axis either exceeds the clamp value or is too small. The specified rotation speed ratio thus cannot be maintained. For polygon turning with two spindles: More information as to why this alarm occurred is indicated in diagnosis data No. 471.

Caution

⚠ **CAUTION**

1. The maximum spindle speed for each gear stage (Parameters No. 3741 to 3744) must be specified correctly according to the model of the machine. In addition, ordinary spindle connections must have been terminated.
2. This function uses the one-rotation signal for the spindle as a reference point for phase adjustment.
   When a built-in sensor is used, and there are gears between the spindle and spindle motor, it is necessary to install a detector on the spindle separately to take a one-rotation signal from the spindle. If the detector does not guarantee detection of a correct position from arbitrary speed, set bit 4 (HST) of parameter No. 7602 and bit 7 (RFCHK3) of serial spindle parameter No. 4016 to 1. This setting reduces the spindle speed automatically down to 0 for spindle position detection each time the polygon turning mode with two spindles is entered, thus guaranteeing a correct phase relationship during polygon synchronization mode with two spindles.
3. This function uses the spindle synchronization function for serial spindles. (However, it does not require the spindle synchronization for the CNC.) You may need to specify the relevant serial spindle parameters (such as Nos. 4032 to 4035 and Nos. 4065 to 4068).
   Specify the same serial spindle loop gain (parameters Nos. 4065 to 4068) for both spindles.
   If the same serial spindle loop gain is not used for both spindles, polygon turning may not be accurate.
   If an attempt is made to perform polygon turning with two spindles using different loop gain, bit 6(LGE) of diagnosis data No.470 becomes 1. (No alarm is issued.)
4. Before using the polygon synchronization mode, place both master axis and polygon synchronization axis in the spindle control mode. If the polygon synchronization mode starts during another mode (Cs contour control mode, spindle orientation, or the like), an alarm occurs.

⚠ **CAUTION**

5 During the polygon synchronization mode, no command cannot be executed for the polygon synchronization axis.
During the polygon synchronization mode, the spindle orientation function cannot be used for either the master axis and polygon synchronization axis. Therefore, gear, tool or workpiece change is basically unusable during the polygon synchronization mode.

6 During polygon synchronization mode, speed change and phase adjustment are performed each time the spindle speed is changed. Therefore, this mode cannot be used together with a function that causes continuous spindle speed change (such as G96 constant surface speed control)

7 During the polygon synchronization mode, the rotation ratio between the master axis and polygon synchronization axis is controlled with priority. Therefore, the difference between the master axis speed and S command value may become larger than during ordinary spindle control.

Note

NOTE

During the polygon synchronization mode, phase control is performed in the least command increment of 360/4096 = 0.08789...(degrees) in reference to the one-rotation signal for each spindle. A setting less that the least command increment is actually invalid.
Since phase control covers relative phase of both spindles within a turn, a setting greater than 360 degrees has no meaning.
This function does not limit the range of valid R settings. A setting greater than 360 degrees is assumed as the remainder of division by 360.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Polygon turning with two spindles

6.9.3 Concurrent Use of Polygon Turning and Polygon Turning with Two Spindles

Overview

When both of polygon turning and polygon turning with two spindles are specified, which polygon turning option is used for each path can be selected using a parameter.

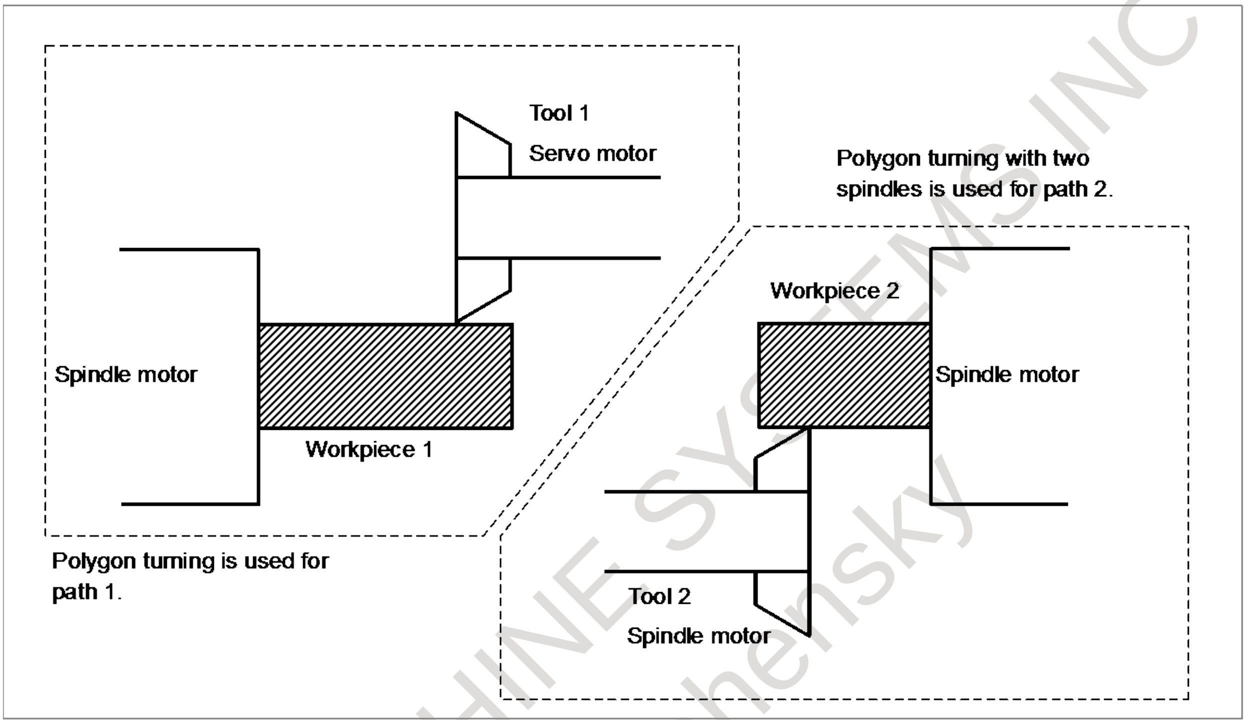


Fig. 6.9.3 (a) Example of performing polygon turning for path 1 and polygon turning with two spindles for path 2

Explanation

- Setting parameters

For a path for which bit 7 (PCG) of parameter No. 7604 is set to 1, whether to perform polygon turning or polygon turning with two spindles can be selected by setting parameter No. 7605. For a path for which bit 7 (PCG) of parameter No. 7604 is set to 0, only polygon turning with two spindles can be performed.

- Alarm

If the parameter for selecting the type of polygon turning is rewritten in the polygon turning mode, alarm PS0314, "ILLEGAL SETTING OF POLYGONAL AXIS", occurs. If an attempt is made to use a spindle used for polygon turning also used for polygon turning with two spindles, alarm PS0314 occurs.

Example)

When polygon turning is performed in path 1, if polygon turning with two spindles with using the same spindle is specified from path 2, alarm PS0314, "ILLEGAL SETTING OF POLYGONAL AXIS", occurs.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7604	PCG							

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#7 PCG If both polygon turning with two spindles and polygon turning are specified:
0: Polygon turning with two spindles is performed.
1: Either of the options is enabled depending on the setting of parameter No. 7605.

7605	Polygon turning type selection

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0, 1

If both polygon turning with two spindles and polygon turning are specified, this parameter can be used to select one of the options for use. A type of polygon turning is selected according to the setting as follows:
0: Polygon turning with two spindles
1: Polygon turning
If a value other than 0 or 1 is specified, 0 is assumed.

> **NOTE**
> Before re-setting this parameter, using the PMC window function or the G10 command, cancel polygon turning (G50.2). In addition, when re-setting this parameter, using the PMC window function, use the M code preventing buffering.

Alarm and message

Number	Message	Description
PS0314	ILLEGAL SETTING OF POLYGONAL AXIS	An axis was specified invalidly in polygon turning. For polygon turning: A tool rotation axis is not specified. (Parameter No. 7610) For polygon turning with two spindles: Valid spindles are not specified. (Parameter Nos. 7640 to 7643) - A spindle other than the serial spindle. - A spindle is not connected. For concurrent use of polygon turning and polygon turning with two spindles: - In the polygon turning mode, the value of parameter No. 7605 (selecting the type of polygon turning) was changed. - An attempt is made to use a spindle used for polygon turning also for polygon turning with two spindles.

6.10 NORMAL DIRECTION CONTROL

Overview

When a tool with a rotation axis (C-axis) is moved in the XY plane during cutting, the normal direction control function can control the tool so that the C-axis is always perpendicular to the tool path (Fig. 6.10 (a): Sample Movement of the tool).

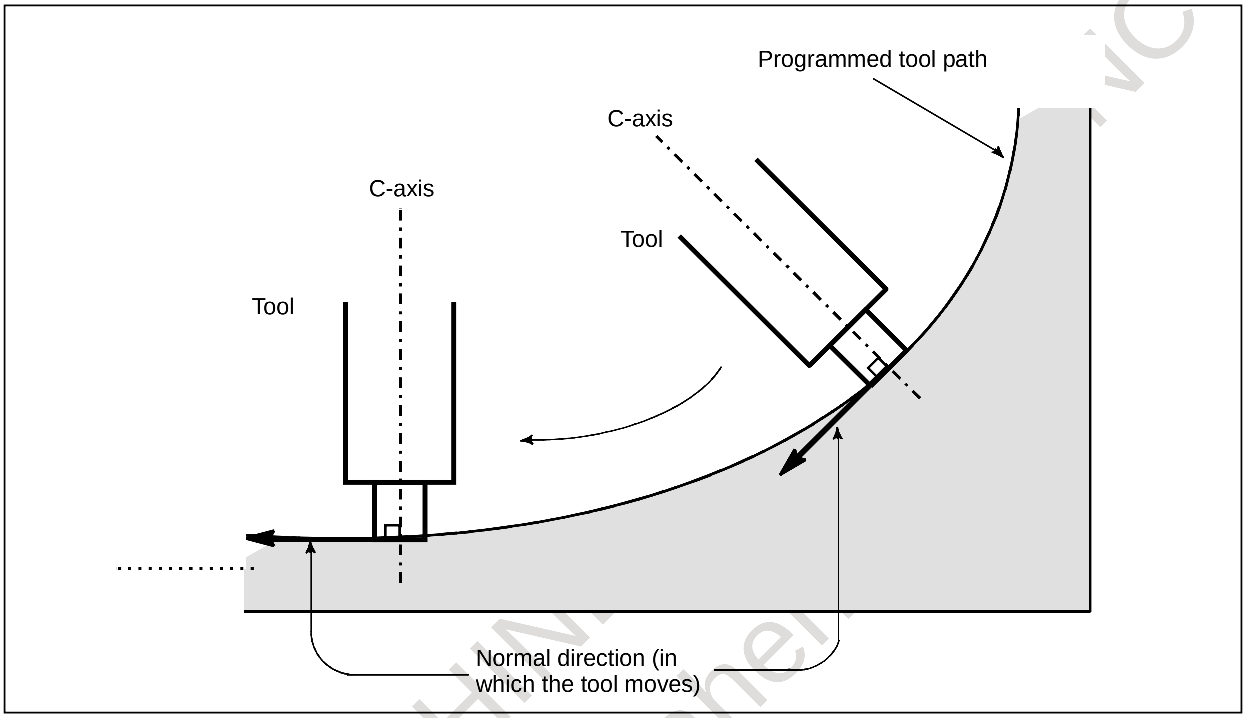


Fig. 6.10 (a) Sample Movement of the tool

Movement of the tool inserted at the beginning of each block is executed at the feedrate set in parameter No. 5481. If dry run mode is on at that time, the dry run feedrate is applied. If the tool is to be moved along the X-and Y-axes in rapid traverse (G00) mode, the rapid traverse rate is applied.
If the feedrate of the C axis exceeds the maximum cutting feedrate of the C axis specified to parameter No. 1430, the feedrate of each of the other axes is clamped to keep the feedrate of the C axis below the maximum cutting feedrate of the C axis.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx
#1 ROSx Setting linear or rotary axis.

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No. 1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

NOTE
Just a rotation axis can be subjected to normal direction control.

5480	Number of the axis for controlling the normal direction

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to the maximum controlled axis number

This parameter sets the controlled axis number of the axis for controlling the normal direction.

5481	Feedrate of rotation of the normal direction controlled axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg/min
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)

This parameter sets the feedrate of the movement along the normal direction controlled axis that is inserted at the start point of a block during normal direction control.

5482	Limit value used to determine whether to ignore the rotation insertion of the normal direction controlled axis

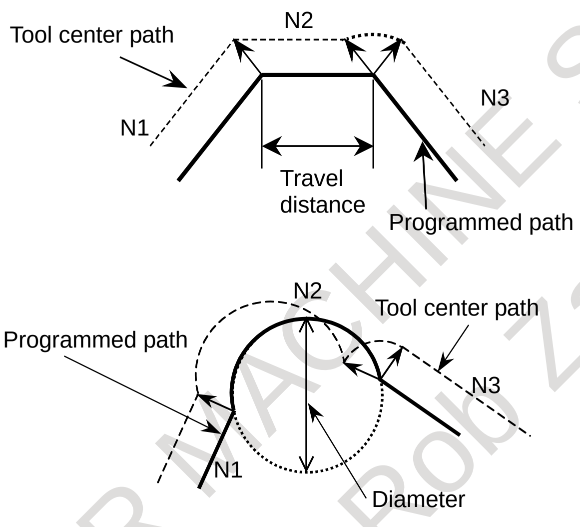
[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
The rotation block of the normal direction controlled axis is not inserted when the rotation insertion angle calculated during normal direction control does not exceed this setting.
The ignored rotation angle is added to the next rotation insertion angle, and the block insertion is then judged.

> **NOTE**
> 1 No rotation block is inserted when 360 or more degrees are set.
> 2 If 180 or more degrees are set, a rotation block is inserted only when the circular interpolation setting is 180 or more degrees.

5483	Limit value of movement that is executed at the normal direction angle of a preceding block

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to standard parameter setting table (B)



For straight line
When the travel distance of N2 in the figure on the left does not exceed the setting, block N2 is machined with the tool being normal to block N1.

For arc
When the arc diameter of N2 in the figure on the left does not exceed the setting, arc N2 is machined with the tool being normal to block N1. A normal direction axis is not controlled to move in the normal direction according to the arc movement.

Fig. 6.10 (b)

1430	Maximum cutting feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Specify the maximum cutting feedrate for each axis.
The cutting feed rate is clamped so that the feedrate in the tangent direction does not exceed the parameter setting.

Alarm and message

Number	Message	Description
PS1470	G40.1 –G42.1 PARAMETER MISS	A parameter setting related to normal direction control is illegal. The axis number of a normal direction controlled axis is set in parameter No. 5480, but that axis number is in the range of the number of controlled axes. The axis set as a normal direction controlled axis is not set as a rotation axis (bit 0 (ROTx) of parameter No. 1006) = 1 and No.1022=0).
PS1471	ILLEGAL COMMAND IN G40.1 –G42.1	An invalid G code was specified in the normal direction control mode.

Limitation

NOTE

1 Do not specify any command to the C axis during normal direction control. Any command specified at this time is ignored.
2 Before processing starts, it is necessary to correlate the workpiece coordinate of the C axis with the actual position of the C axis on the machine using the coordinate system setting (G92) or the like.
3 Helical cutting cannot be specified in the normal direction control mode.
4 Normal direction control cannot be performed by the G53 move command.
5 The C-axis must be a rotation axis.
6 The following functions must be commanded with normal direction control cancel mode.
 - Plane selection command
 - Spiral interpolation
 - Conical interpolation
 - AI contour control

T

NOTE

In the normal direction control mode, the following commands cannot be issued. An attempt to issue any of them results in alarm PS1471 being raised.
- Plane selection command (G17, G18, G19)
- Automatic reference position return(G28)
- 2nd/3rd/4th reference position return(G30)
- Move from the reference position (G29)

6.11 LINEAR INTERPOLATION (G28, G30, G53)

Overview

When positioning operation of linear interpolation type is specified (bit 1 (LRP) of parameter No. 1401 = 1), the following operations can also be set as operations of linear interpolation type by setting bit 4 (ZRL) of parameter No. 1015 to 1:

- Movement from an intermediate point to a reference position in automatic reference position return operation (G28)
- Movement from an intermediate point to a reference position in second, third, or fourth reference position return operation (G30)
- Positioning by machine coordinate system selection (G53)

When any of the operations above is set as an operation of linear interpolation type, acceleration/deceleration control follows the setting of bit 4 (PRT) of parameter No. 1603.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1015				**ZRL**				

[Input type] Parameter input
[Data type] Bit path

#4 ZRL When a reference position is established, the tool path from the middle point to the reference position and machine coordinate positioning (G53) in automatic reference position return (G28) or 2nd/3rd/4th reference position return (G30) are based on:
0: Positioning of nonlinear interpolation type
1: Positioning of linear interpolation type

> **NOTE**
> This parameter is valid when bit 1 (LRP) of parameter No. 1401 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1603				**PRT**				

[Input type] Parameter input
[Data type] Bit path

#4 PRT For positioning of linear interpolation type:
0: Acceleration/deceleration of acceleration fixed type is used.
1: Acceleration/deceleration of time fixed type is used.

Note

> **NOTE**
> Automatic reference position return operation of low-speed type (G28)
> If reference position return operation is not performed for a specified axis even once after the power is turned on in automatic reference position return operation (G28), a reference position return operation of low-speed type is performed for the axis.
> At this time, positioning of non-linear interpolation type is performed.
> For example, if G28X0Y0Z0; is specified when reference position return operation is completed for the X-axis and Y-axis, and is not completed for the Z-axis, positioning of non-linear interpolation type is performed.

6.12 SMOOTH TOLERANCE+ CONTROL

M

Overview

Smooth tolerance+ control is a function that generates smooth machining path within specified tolerance. In the case of die mold machining program that consists of small line segments, this function acts as smoothing function and makes joints of blocks smooth. As the result, machining surface gets high quality.

Smoothing small line segments

This function smooths small line segments. If the length of linear blocks is less than the value set at parameter No. 19595, those blocks are treated as small line segment blocks, and smooth curves not depending on commanded points are generated, which are within tolerance from commanded points. Machining gets smoother even if there are small discontinuous blocks in a program.
Tolerance is set as same value as CAM tolerance which generating small line segments. Tolerance is specified by parameters (No.11786, 11787) or NC program (G10.8L4Q_R_).

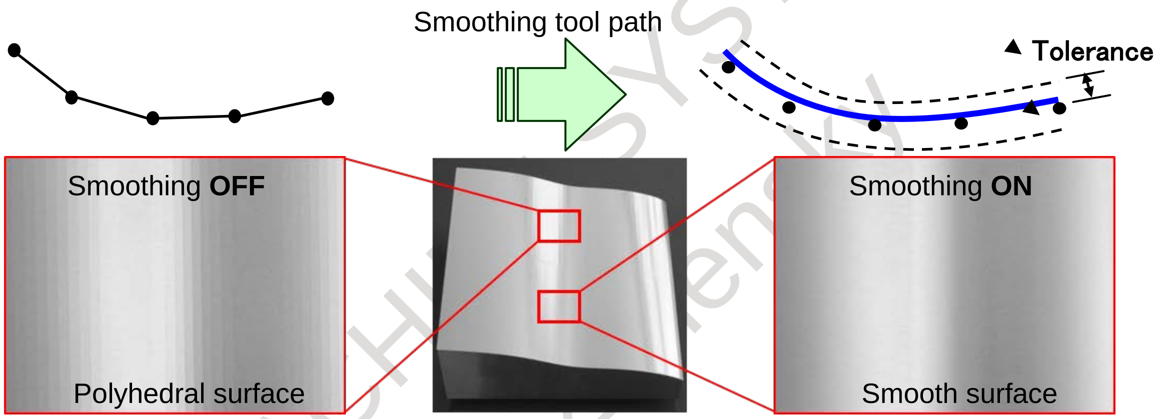


Fig. 6.12 (a) Smoothing small line segments

Command format (In Case of 3-axis Machining)

G05.1 Q3 Xp0 Yp0 Zp0;	**Smooth tolerance+ control mode on**
G10.8 L4 Q_;	**Specify tolerance**
:	
G05.1 Q0;	**Smooth tolerance+ control mode off**

Xp: X-axis or an axis parallel to the X-axis
Yp: Y-axis or an axis parallel to the Y-axis
Zp: Z-axis or an axis parallel to the Z-axis
Q: Tolerance for linear axis on curves

NOTE
1 Specify G05.1 Q3 alone in a block.
(To avoid specifying any other G code in the same block)
2 Unit of "Q" in G10.8 command depends on the increment system of the basic axis.
3 The value of parameter No. 11786 is effective as tolerance between G05.1 Q3 command and G10.8 L4 command.

4 Smooth tolerance^{+} control is disabled when the tolerance for linear axis is set to 0.
5 Smooth tolerance^{+} control mode is activated at the start of automatic operation by setting bit 0 (CAT) of parameter No. 11785.

Description of the Function

- Automatically turning on AI contour control with Smooth tolerance^{+} control

Specifying G5.1 Q3 also enables Smooth tolerance^{+} control and AI contour control to be turned on at the same time.
The automatic velocity control by AI contour control reduces impacts on the mechanical system.

- Conditions for enabling Smooth tolerance^{+} control

Smooth tolerance^{+} control is enabled when the following conditions are satisfied.
In a block that does not satisfy the conditions for enabling it, Smooth tolerance^{+} control is canceled, and it is judged
in the next block whether to perform Smooth tolerance^{+} control again.

- The mode is all of the following:
  - Cutting mode (G64)
  - Linear interpolation (G01)
  - Feed per minute (G94, however G98 in G code system A of T series)
    or Feed per revolution(G95, however G99 in G code system A of T series)
  - Constant surface speed control cancel (G97)
  - Normal direction control cancel (G40.1)
  - Polar coordinate command cancel (G15)
  - Polar coordinate interpolation cancel (G13.1)
  - Polygon turning cancel (G50.2)
- The block does not contain a one shot G code command.
- The block does not contain MST command.
- The block does not suppress look ahead (buffering).
- The block contains a move for at least a linear axis.
- The block contains a move command for only axes subject to Smooth tolerance^{+} control.
- The difference between a radius at the beginning point and a radius at the end point is less than 20µm.
- Smooth tolerance^{+} control disable signal STCD <Gn599.2> is “0”.
- Block length is less than the value of parameter No. 19595.

- Interlock

When an axis for Smooth tolerance^{+} control is interlocked, all axes are interlocked in the block which Smooth tolerance^{+} control is enabled, even if the interlocked axis is not commanded in the block.

- Axis moving signals

In the block which Smooth tolerance^{+} control is enabled, axis moving signals MV1 to MV8 <Fn102> for axes for Smooth tolerance^{+} control are set to one regardless of the movement.

- Use with other functions

In the case Smooth tolerance^{+} control is used with the following functions, it controls paths which each function is applied.

- Cutter compensation and tool nose radius compensation
- Tool length compensation
- Programmable mirror image
- Scaling
- Coordinate system rotation

Use with cutter compensation and tool nose radius compensation

In case using cutter compensation and tool nose radius compensation, Smooth tolerance+ control works for paths cutter compensation and tool nose radius compensation applied.

- Status display in the Smooth Tolerance+ control mode

Character string “TOLCON” blinks in the status bar when Smooth tolerance+ control mode is enabled.

Fig. 6.12 (b) Status display in the Smooth Tolerance+ control mode

6.12.1 Change Tolerance in Smooth Tolerance+ Control Mode

The tolerance in smooth tolerance+ control mode can be changed any time by specifying G10.8 L4. The appropriate tolerance can be used which depends on situations in a program.

Format

Change Tolerance in Smooth Tolerance+ Control Mode

G10.8 L4 Q_ R_;	Directly specify tolerance
G10.8 L4;	Use paramter (No.11786, No.11787) as default tolerance

Q: Tolerance for linear axis on curves
R: Tolerance for rotary axis on curves

NOTE

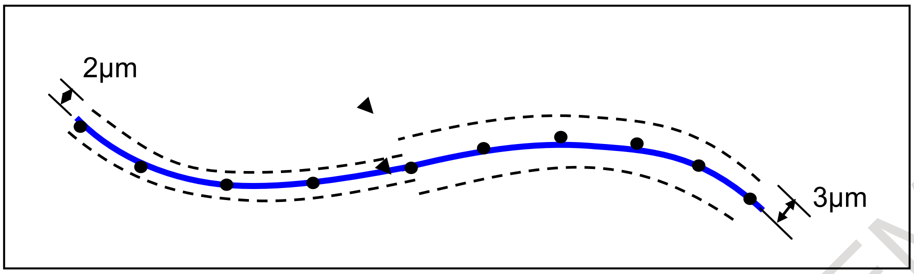
1. Specify G10.8 alone in a block. (Avoid specifying any other G code in the same block)
2. G10.8 L4 is a one-shot G code.
3. Unit of “Q” and “R” in G10.8 command depends on the increment system of the basic axis.
4. Specifying a negative value to “Q” or “R” causes alarm PS2010, “ILL. COMMAND IN TOLERANCE CON.”.
5. When G10.8 L4 specified not in smooth tolerance+ control mode (G05.1 Q3), alarm PS0412, “ILLEGAL G CODE” is issued.
6. The value of parameter No. 11786 and No. 11787 do not change by specifying G10.8 L4.

Example (1)

Here is an example for changing tolerance in smooth tolerance+ control mode.

```
O0012;
G28 G91 X0 Y0;
G05.1 Q3 X0 Y0 Z0;              (Smooth tolerance+ control mode on)
G10.8 L4 Q0.002;                (Tolerance is set.)
G00 G90 X100.0 Y100.0 Z100.0;
G01 X100.121 Y100.321 F1000;
X100.121 Y100.321;
X100.321 Y100.881;
X100.721 Y101.365;              Smoothing (Tolerance = 2μm)
  :
  :
```

```
G10.8 L4 Q0.003;          (Tolerance is changed.)
X110.121 Y120.321;  ┐
X110.321 Y120.881;  │
X110.721 Y121.365;  ├  Smoothing (Tolerance = 3µm)
   :                │
   :                ┘
G05.1 Q0;                 (Smooth tolerance+ control mode off)
M30;
```



Signal

Smooth tolerance^{+} control disable signal STCD <Gn599.2>

[Classification] Input signal

[Function] Switches Smooth tolerance^{+} control mode on / off.

[Operation] When a signal is set to "1", Smooth tolerance^{+} control is disabled.
This signal is effective for blocks analyzed after the signal switched.

> **Note**
> 1 Do not change this signal in the feed hold stop state. If this signal is changed, alarm PS2012 "ILL. OPERATION IN TOLERANCE CON." is issued in restarting operation.
> 2 Switching this signal does not mean switching Smooth tolerance^{+} control mode immediately. Actual switch takes place when Smooth tolerance^{+} control mode can be changed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn599						STCD		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11785								CAT

[Input type] Parameter input
[Data type] Bit axis

#0 CAT On start up of automatic operation, smooth tolerance^{+} control is:
0: Ineffective on an axis.
1: Effective on an axis.

11786	Smoothing tolerance for linear axis in smooth tolerance^{+} control mode

[Input type] Parameter input
[Data type] Real path

[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.000 to +999999.999)
Set tolerance for linear axes for curves generated in smooth tolerance^{+} control mode. If tolerance is not specified in tolerance mode, machining path is controlled so that the machining accuracy on curves represented by small segment is within the tolerance set to this parameter.
When this parameter is set to 0 or less, the smoothing is not effective.

11787	Smoothing tolerance for rotary axis in smooth tolerance^{+} control mode

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.000 to +999999.999)
Set tolerance for rotary axes for curves generated in smooth tolerance^{+} control mode. If tolerance is not specified in tolerance mode, machining path is controlled so that the machining accuracy on curves represented by small segment is within the tolerance set to this parameter.
When this parameter is set to 0 or less, it operates as 0.05° is set.

	#7	#6	#5	#4	#3	#2	#1	#0
11790					SMC			

[Input type] Parameter input
[Data type] Bit path

#3 SMC When smooth tolerance^{+} control is used with macro modal call, smooth tolerance^{+} control is,
0: Effective.
1: Disabled. (Temporary disabled while macro modal call is effective)

11791	Angle for breaking smoothing in smooth tolerance^{+} control mode

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to 90
Set the angle to judge if smoothing of smooth tolerance^{+} control is effective or not.
When this parameter is set to 0, the smoothing is not disabled.

19595	Maximum block length for small line segments in smooth tolerance^{+} control mode

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.000 to +999999.999)
Set maximum block length for regarding a linear block as a small line segment in smooth tolerance+ control mode. If the block length is less than the value of this parameter, smoothing of small line segments is applied to the block.
When this parameter is set to 0 or less, smoothing is applied to any block.

19599	Maximum allowable acceleration rate for the deceleration function based on acceleration in smooth tolerance+ control mode for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec2, inch/sec2, deg/sec2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Feedrate is controlled so that acceleration produced by changing the move direction does not exceed the value specified to this parameter in smooth tolerance+ control mode.
This parameter is valid for basic 3-axes (the axes that 1, 2, 3 are set in the parameter No.1022). This parameter of the other axes is invalid, and not necessary to be set.
For an axis with 0 set in this parameter, the deceleration function based on acceleration in smooth tolerance+ control mode is disabled.
If a different value is set to this parameter for each axis, a feedrate is determined from the smallest of the acceleration rates specified for the moving axes.

Diagnosis data

5000	The index number of the spindle axis that synchronizes with each servo axis

[Data Type] Bit
TOLERANCE ON

When smooth tolerance+ control G5.1 Q3 is specified and all conditions are satisfied, "1" is indicated. G5.1 Q3 command turns on AI contour control at the same time. Therefore, the AI contour control mode signal AICC<Fn062.0> turns on.

5006	Tolerance for linear axes for curves generated in smooth tolerance+ control mode

[Data type] Real path
[Unit of data] mm/inch
[Valid data range] 0 or positive 9 digit of minimum unit of data
[Meaning] Tolerance for linear axes for curves generated in smooth tolerance+ control mode is displayed.
In the beginning of smooth tolerance+ control mode, the value of a parameter No.11786 is displayed.
When the tolerance is changed by G10.8L4, the changed tolerance is displayed.

5007	Tolerance for rotary axes for curves generated in smooth tolerance+ control mode

[Data type] Real path
[Unit of data] deg
[Valid data range] 0 or positive 9 digit of minimum unit of data
[Meaning] Tolerance for rotary axes for curves generated in smooth tolerance+ control mode is displayed.
In the beginning of smooth tolerance+ control mode, the value of a parameter No.11787 is displayed.
When the tolerance is changed by G10.8L4, the changed tolerance is displayed.

Alarm

Number	Message	Description
PS0412	ILLEGAL G CODE	Illegal G code is specified. - G10.8L4 is specified not in smooth tolerance+ control mode.
PS2010	ILL. COMMAND IN TOLERANCE CON.	Illegal command is specified in smooth tolerance+ control mode. Check the program.
PS2011	ILL. PARAMETER IN TOLERANCE CON.	Illegal parameter is set, which is related to smooth tolerance+ control. Check the parameter settings.
PS2012	ILL. OPERATION IN TOLERANCE CON.	Illegal operation is done in smooth tolerance+ control mode.

Note

- Functions Which Smooth tolerance$^+$ control Is Temporary Disabled

This function is temporary disabled when the following functions are used together.

Functions
Any kind of interpolations except liner interpolation.
Inverse time feed
Constant surface speed control cancel
Normal direction control
Polar coordinate command
Polar coordinate interpolation
Polygon turning
Tool length compensation command block
Canned cycle

- Background graphic display

The background graphic display function draws the path in Smooth tolerance+ control mode by linear interpolation.

- Functions Which Does Not Work with Smooth Tolerance$^+$ Control

This function doesn't work with the following functions.

Functions	Alarm (1)
Interruption type custom macro	PS2012

(1) Functions which issue alarms in case using with Smooth tolerance$^+$ control.

6.13 GENERAL PURPOSE RETRACT

Overview

In automatic operation mode or in manual operation mode, setting the retract signal RTRCT <Gn066.4> to 1 causes this function to capture the rise of this signal, causing the tool to move (retract) along the axis for which a retract amount is specified for parameter No. 7741. After the end of retraction, the retract completion signal RTRCTF <Fn065.4> is output. This function is intended to retract the tool from the workpiece immediately when a tool breakage is detected.

- The feedrate assumed during retract is the same as that set in parameter No. 7740. A feedrate override is invalid.
- A feed hold is invalid to movement during retraction.
- If the retract signal is set to 1 during automatic operation, a retract operation is performed and automatic operation is stopped.
- The retract completion signal becomes 0 when the tool has moved along one of the retract axes.

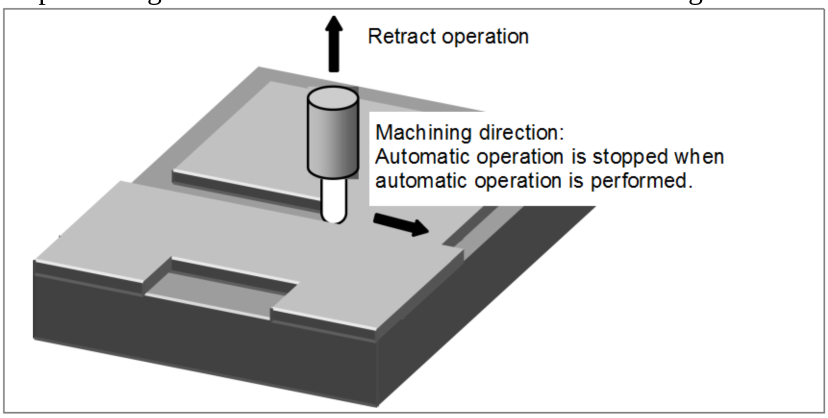


Fig. 6.13 (a)

Signal

Retract signal RTRCT<Gn066.4>

[Classification] Input signal

[Function] Causes the tool to retract along the axis for which a retract amount is set in the parameter.

[Operation] When this signal becomes 1, the CNC operates as described below.

- Captures the rise of the signal and causes the tool to retract along the axis for which a retract amount is set in parameter No. 7741. The retract amount and the retract speed assume the values previously set in parameters Nos. 7741 and 7740.
  After the end of retraction, the retract completion signal RTRCTF becomes 1.
  The retract signal is effective in either automatic operation mode (MEM, MDI) or manual mode (HNDL, JOG, etc.).
  If the retract signal is set to 1 during automatic operation, a retract operation is performed and automatic operation is stopped.

Retract completion signal RTRCTF<Fn065.4>

[Classification] Output signal

[Function] Notifies that retraction is completed.

[Output cond.] This signal becomes 1 when:

- A retract operation is completed.
  If a retract operation is performed on two or more axes, the operation is completed on all retract axes.

It becomes 0 when:

- After the end of a retract operation, a move command is issued for one of the retract axes.
- Follow-up occurs due to a servo-off, emergency stop, or other conditions.

> **NOTE**
> While the retract completion signal is 1, the retract signal cannot be accepted.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn066				RTRCT				

	#7	#6	#5	#4	#3	#2	#1	#0
Fn065				RTRCTF				

Timing chart

(1) RTRCT and RTRCTF on/off timing

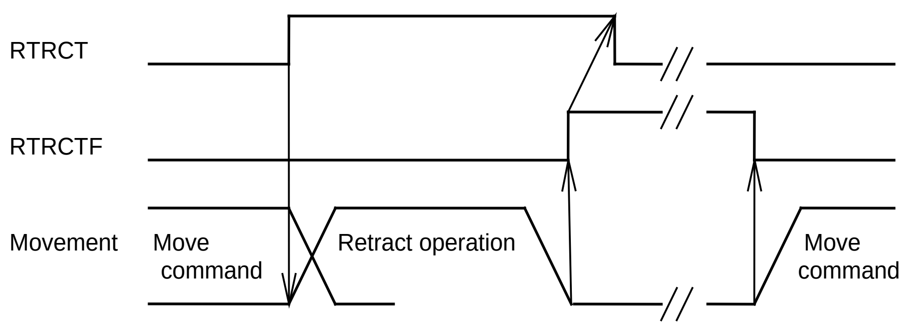
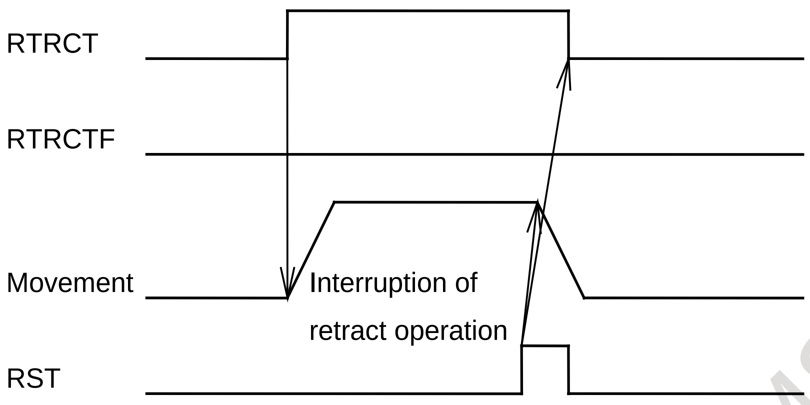


Fig. 6.13 (b)

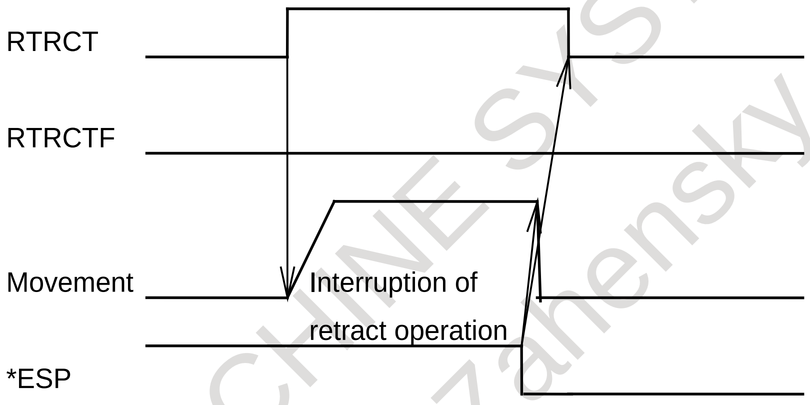
(2) Interruption of retraction due to a reset



Turn the RTRCT signal off at the same time as turning the RST signal on.

Fig. 6.13 (c)

(3) Interruption of retraction due to an emergency stop



Turn RTRCT off at the same time as turning *ESP off.

Fig. 6.13 (d)

Parameter

7740	Feedrate during retraction

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the feedrate during retraction for each axis.

7741	Retract amount

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the retract amount for each axis.

> **NOTE**
> The tool moves (is retracted) by the specified amount regardless of whether diameter or radius programming is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
7704								ACR

[Input type] Parameter input
[Data type] Bit path

#0 ACR In the AI contour control mode, general purpose retract operation is:
0: Not Used.
1: Used.

7745	Time constant for linear acceleration/deceleration in retract operation for each axis

[Input type] Parameter input
[Data type] word axis
[Unit of data] msec
[Valid data range] 0 to 1000

This parameter sets an acceleration rate for linear acceleration/deceleration in retract operation based on the general-purpose retract function. Set a time (Time constant) used to reach the federate set in parameter No.7740 for each axis.

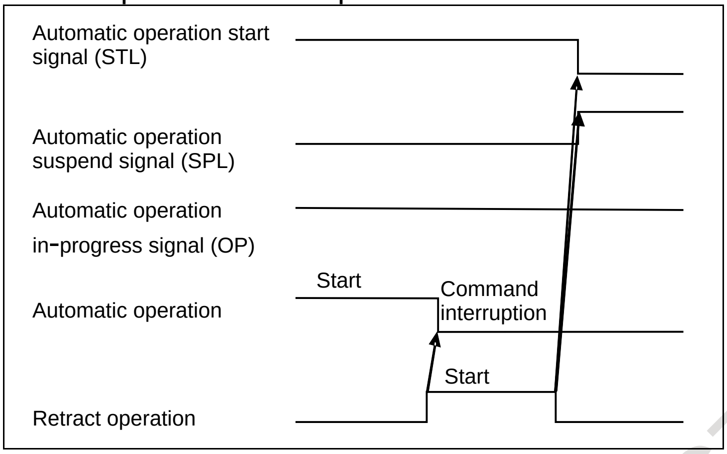
> **NOTE**
> This parameter is valid when bit 0 (ACR) of parameter No. 7704 is set to 1 to perform a retract operation in the AI contour control mode.

Caution

> ⚠ **CAUTION**
> 1 A feedrate override is invalid to the retract speed.
> 2 During a retract operation, an interlock to a retract axis is effective.
> 3 During a retract operation, a machine lock to a retract axis is effective. The retract operation is completed in the machine lock state, and the retract completion signal is output.
> 4 During retraction, a feed hold is invalid.
> 5 The retract direction is the movement direction of the machine regardless of whether a mirrored image (signal and setting) is valid or not. (A mirror image is not applied to the updating of absolute coordinates.)

⚠ CAUTION

6 When retraction is performed during automatic operation, the execution of a command is interrupted as soon as a retract operation starts. However, the operation state is switched to the automatic operation stopped state when the retract operation is completed.



7 Retract movement is effected with non-linear type positioning.

8 During a retract operation, acceleration/deceleration assumes the acceleration/deceleration state at the start of retraction. In AI contour control mode, acceleration/deceleration before interpolation is not applied to a retract operation. By setting 1 in ACR, bit 0 of parameter No. 7704, and setting a retract time constant in parameter No. 7745, it is possible to apply linear acceleration/deceleration to a retract operation in AI contour control mode.

9 During a retract operation, performing a reset or emergency stop causes the operation to be interrupted. In this case, the retract completion signal does not become 1.

10 If, during following alarms are generated, the retract operation is interrupted. In this case, the retract completion signal does not become 1.
With alarms other than following alarms, the retract operation is not interrupted
- Servo alarm
- Retract axis OT alarm
- PW alarm
- Alarm DS0001 or DS0002

11 After the start of a retract operation, setting the retract signal RTRCT to 0 does not cause the retract operation to be interrupted.

12 While the retract completion signal RTRCTF is 1, the retract signal RTRCT cannot be accepted.

13 Even while a threading command is being executed, the retract function is effective. When the retract function is executed, the threading operation is stopped immediately, and a retract operation is started.

14 Even in a canned cycle (during tapping), the retract function is effective. When the retract function is executed, the cycle operation is stopped immediately, and a retract operation is started.

15 When retraction is performed on an axis on which movement is effected with PMC axis control, the move command for the PMC axis is not canceled. If, therefore, retraction is to be performed on a PMC axis on which movement is effected, set the retract signal RTRCT to 1 and use the PMC axis reset signal ECLRg to cancel PMC axis control.

⚠ **CAUTION**

16 If an electronic gear box is attached, this function is invalid.

17 If a group that has selected flexible synchronization control wishes to perform a retract operation when a sync mode is valid, the group must set the retract function for the master axis only. If the retract function is set for the slave axis, a retract operation is not performed normally.

18 If general purpose retract is executed on an axis for which axis synchronous control is enabled, the parameters below causes a retract operation to be performed according to the settings of the master axis.
- Feedrate during retraction (parameter No. 7740)
- Retracted distance (parameter No. 7741)
- Time constant during retraction in AI contour control (parameter No. 7745)

Do not perform general purpose retract while the grid positioning automatic setup of axis synchronous control is being executed.

19 This function cannot be used for axes related to 3-dimensional coordinate system conversion.

20 When this function is used in the axis involved with Synchronous, Composite, or Superimposed control, alarm DS1934, “RETRACT CANNOT BE USED” is issued. Retract operation is not performed in the path including this axis.

7 FEEDRATE CONTROL/ACCELERATION AND DECELERATION CONTROL

7.1 FEEDRATE CONTROL

The feed functions control the feedrate of the tool. The following two feed functions are available:

1. Rapid traverse
   When the positioning command (G00) is specified, the tool moves at a rapid traverse rate set in the CNC (parameter No. 1420).
2. Cutting feed
   The tool moves at a programmed cutting feedrate.

Override can be applied to a rapid traverse rate or cutting feedrate using the override signal.

7.1.1 Rapid Traverse Rate

Overview

The positioning command (G00) positions the tool by rapid traverse.

G00 IP_ ;

G00 : G code (group 01) for positioning (rapid traverse)
IP_ : Dimension word for the end point

In rapid traverse, the next block is executed after each axis stops at the end point and the servo motor reaches a certain range set by the parameter No. 1826 (in-position check).
A rapid traverse rate is set for each axis by parameter No. 1420 or No. 1428 (before reference position establishment), so no rapid traverse rate need be programmed.
The following overrides can be applied to a rapid traverse rate with the rapid traverse override signals ROV1, ROV2<Gn014.0, 1>: F0, 25, 50, 100%
F0: Allows a fixed feedrate to be set for each axis by parameter No. 1421.
The 1% step rapid traverse override selection signal HROV<Gn096.7> or 0.1% step rapid traverse override selection signal FHROV<Gn353.7> enables a rapid traverse override to be applied in steps of 1% or 0.1% over a range of 0% to 100%.
In case that bit 4 (RF0) of parameter No. 1401 is set to 1, the machine tool stops moving when cutting feedrate override is 0% during rapid traverse.
Positioning (G00) is performed with either non-linear or linear interpolation can be specified by bit 1 (LRP) of parameter No. 1401.
Dry run for rapid traverse command is enabled by setting bit 6 (RDR) of parameter No. 1401 to 1.
For details of override, refer to “Rapid traverse override”.
For details of manual rapid traverse, refer to “JOG FEED/INCREMENTAL FEED”.

Signal

Rapid traversing signal RPDO<Fn002.1>

[Function] This signal indicates that a move command is executed at rapid traverse.
[Output cond.] 1 indicates that an axis starts moving after rapid traverse has been selected in automatic operation mode or manual operation mode.
0 indicates that an axis starts moving after a feedrate other than rapid traverse has been selected in automatic operation mode or manual operation mode.

NOTE

1 The rapid traverse in automatic operation includes all rapid traverses in canned cycle positioning, automatic reference point return, etc., as well as the move command G00. The manual rapid traverse also includes the rapid traverse in reference position return.

2 This signal may be 1 when not only during axis movement but also when the axis is stopped.Once rapid traverse has been selected and movement is started, this signal remains 1, including during a stop, until another feedrate has been selected and movement is started.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn002							RPDO	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401		RDR		RF0			LRP	

[Input type] Parameter input
[Data type] Bit path

#1 LRP Positioning (G00)
0: Positioning is performed with non-linear type positioning so that the tool moves along each axis independently at rapid traverse.
1: Positioning is performed with linear interpolation so that the tool moves in a straight line.
When using 3-dimensional coordinate system conversion, set this parameter to 1.

#4 RF0 When cutting feedrate override is 0% during rapid traverse,
0: The machine tool does not stop moving.
1: The machine tool stops moving.

#6 RDR Dry run for rapid traverse command
0: Disabled
1: Enabled

1420	Rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1421	F0 rate of rapid traverse override for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)

[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the F0 rate of the rapid traverse override for each axis.

1826	In-position width for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
The in-position width is set for each axis.
When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position. (The machine is in the in-position state.)

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Rapid traverse

7.1.2 Cutting Feedrate Clamp

Overview

An upper limit can be set on the cutting feedrate along each axis (parameter No. 1430). If an actual cutting feedrate (with an override applied) exceeds a specified upper limit, it is clamped to the upper limit.

Parameter

1430	Maximum cutting feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Specify the maximum cutting feedrate for each axis.

Alarm and message

Number	Message	Description
PS5009	PARAMETER ZERO (DRY RUN)	The dry run rate parameter No. 1410 or the parameter for the maximum cutting feedrate for each axis is 0. The parameter for the maximum cutting feedrate for each axis is No. 1432 if look-ahead acceleration/deceleration before interpolation is enabled and No. 1430 otherwise. Functions that cause look-ahead acceleration/deceleration before interpolation include AI contour control.

Number	Message	Description
PS5011	PARAMETER ZERO (CUT MAX)	The setting of the parameter for the maximum cutting feedrate is 0. The parameter is No. 1432 if look-ahead acceleration/deceleration before interpolation is enabled and No. 1430 otherwise. Functions that cause look-ahead acceleration/deceleration before interpolation include AI contour control.

Warning

⚠ WARNING

CNC calculation may involve a feedrate error of ±2% with respect to a specified value. However, this is not true for acceleration/deceleration. To be more specific, this error is calculated with respect to a measurement on the time the tool takes to move 500 mm or more in the steady state.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Cutting feed

7.1.3 Feed per Minute

Overview

- Feed per minute

After specifying G94 (G98 for G code system A in lathe system) (in the feed per minute mode), the amount of feed of the tool per minute is specified by setting a number after F. G94 (G98 for G code system A in lathe system) is a modal code. Once a G94 (G98 for G code system A in lathe system) is specified, it is valid until another G code belonging to the group 05, such as G95 (G99 for G code system A in lathe system) (feed per revolution) is specified. At power-on, the feed per minute mode (feed per revolution mode for lathe system and bit 4(FPM) of parameter No.3402 is set to 0) is set.

An override from 0% to 254% (in 1% steps) can be applied to feed per minute with the feedrate override signals *FV0 to *FV7 <Gn012>.

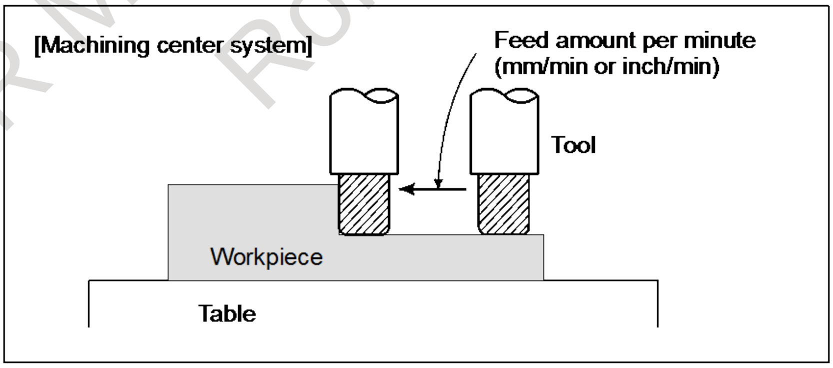


Fig. 7.1.3 Feed per minute

⚠ CAUTION

No override can be used for any commands such as for threading.

Format

M

Feed per minute

G94; **G code for feed per minute (Group 05)**
F_; **Feed rate (mm/min or inch/min)**

T

G code system A

Feed per minute

G98; **G code for feed per minute (Group 05)**
F_; **Feed rate (mm/min or inch/min)**

G code system B/C

Feed per minute

G94; **G code for feed per minute (Group 05)**
F_; **Feed rate (mm/min or inch/min)**

Note

- Feedrate (F) at the switching of feed mode

If a modal code in group 05 is switched with a feed selection command, the feedrate (F) is inherited as modal information, which means that cutting may be performed at an unintended feedrate. If switching a modal code in group 05, be sure to specify a feedrate (F).
If bit 4 (MFC) of parameter No. 13450 is set to 1, the feedrate (F) is cleared if a modal code in group 05 is switched. This causes alarm PS0011, "FEED ZERO (COMMAND)", to be issued. By inheriting the feedrate (F) as modal information, it is possible to prevent cutting from being performed at an unintended feedrate.

Example)

Execute the program below by setting bit 4 (MFC) of parameter No. 13450 to 1.
At the block for switching from G94 to G95 (N06), alarm PS0011 is issued.

```
O0001 ;
N01 G90 G00 X0.0 Y0.0 ;
N02 M03 S100 ;
N03 G91 G01 ;
N04 G94 X10.0 F100.0 ;
N05 Y10.0 ;
N06 G95 X10.0 ; ⇒ Alarm PS0011 is issued.
N07 Y10.0;
```

M

Feed selection commands are the following G codes in group 05.
G93 (inverse time feed) /G94 (feed per minute) / G95 (feed per revolution)

T

Feed selection commands are the following G codes in group 05.
G code system A
G93 (inverse time feed) /G98 (feed per minute) /G99 (feed per revolution)

G code system B/C

G93 (inverse time feed) /G94 (feed per minute) /G95（ (feed per revolution)

- Increment system of F command

The increment system of an F command in the feed per minute mode is as follows:

M

Table 7.1.3 (a) Increment system of F command (for the machining center system)

Decimal point	Metric input	Inch input
With decimal point	Depend on the increment system of the reference axis (IS-A, B, C)	
With no decimal point	1 mm/min	0.01 inch/mm

T

Table 7.1.3 (b) Increment system of F command (for the lathe system)

Decimal point	Bit 2 (FM3) of parameter No.1404	Metric input	Inch input
With decimal point	-	Depend on the increment system of the reference axis (IS-A, B, C)	
With no decimal point	0	1 mm/min	0.01 inch/min
	1	0.001 mm/min	0.00001 inch/min

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
13450				MFC				

[Input type] Parameter input
[Data type] Bit path

#4 MFC When the cutting is executed without specifying a feedrate (F) after the modal G code of group 05 was changed by G93(inverse time feed) / G94(feed per minute) / G95(feed per revolution) command,

0: The feedrate (F) is inherited as a modal.
1: Alarm PS0011, “FEED ZERO (COMMAND)” is issued.

NOTE

1 In G93 mode, if the axis command and the feedrate (F) command are not in the same block, alarm PS1202, "NO F COMMAND AT G93" is issued regardless of the setting of this parameter.

2 If this parameter bit is set to 1, and if the G code of group 05 is cleared due to a reset, by setting bit 6 (CLR) of parameter No. 3402 to 1 and bit 5 (C05) of parameter No. 3406 to 0, so that the modal G code is switched, the feedrate (F) will be cleared even if bit 7 (CFH) of parameter No. 3409 is set to 1.

3 If this parameter bit is 1, and bit 7 (FC0) of parameter No. 1404 is set to 1, alarm PS0011 is not issued and the block is executed with a feedrate of 0 even if the feed selection command is used to switch the modal code of group 05 and the axis command is executed in cutting feed mode without specifying a feedrate (F). In G93 mode, alarm PS1202 is issued regardless of the setting of the parameter bit FC0.

NOTE
4 If this parameter bit is 1, alarm PS0011 or PS1202 is not used even if the feed selection command is used to switch the modal code of group 05 and the axis command is executed in cutting feed mode without specifying a feedrate (F), provided that the travel distance is 0.
5 If this parameter bit is 1, alarms PS0011 and PS1202 are issued if the feed selection command is used to switch the modal code of group 05 and the axis command is executed in cutting feed mode without specifying a feedrate (F), even if cutting feedrate (parameter No. 1411) during automatic operation is set. (This is true of the M series.)

	#7	#6	#5	#4	#3	#2	#1	#0
1404	FC0					FM3		
	FC0							

[Input type] Parameter input
[Data type] Bit path

#2 FM3 The increment system of an F command without a decimal point in feed per minute is:
0: 1 mm/min (0.01 inch/min for inch input)
1: 0.001 mm/min (0.00001 inch/min for inch input)

#7 FC0 Specifies the behavior of the machine tool when a block (G01, G02, G03, etc.) containing a feedrate command (F command) that is 0 is issued during automatic operation, as follows:
0: An alarm PS0011, “FEED ZERO (COMMAND)” occurs.
1: An alarm PS0011 does not occur, and the block is executed at feedrate 0.

NOTE
1 In inverse time feed (G93) mode, the alarm PS1202, "NO F COMMAND AT G93" is issued irrespective of the setting of this parameter.
2 This parameter is set from 1 to 0, if a bit 6 (CLR) of parameter No. 3402 is 1, reset the CNC. Or if CLR is 0, turn off and on the CNC.

1411	Cutting feedrate

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

A cutting feedrate can be specified with this parameter for a machine which does not have to change the cutting feedrate frequently during machining. This eliminates the need to specify a cutting feedrate (F code) in the NC program.

	#7	#6	#5	#4	#3	#2	#1	#0
3402		CLR		FPM				
		CLR						

[Input type] Parameter input
[Data type] Bit path

#4 FPM At power-on time or in the cleared state:
0: G99 or G95 mode (feed per revolution) is set.
1: G98 or G94 mode (feed per minute) is set.

#6 CLR RESET key on the MDI unit, external reset signal ERS<Gn008.7>, reset and rewind signal RRW<Gn008.6>, and emergency stop signal
0: Cause reset state.
1: Cause clear state.
For the reset and clear states, refer to Appendix in the OPERATOR'S MANUAL.

	#7	#6	#5	#4	#3	#2	#1	#0
3406			C05					

[Input type] Parameter input
[Data type] Bit

#5 C05 If bit 6 (CLR) of parameter No. 3402 is set to 1, set a G code of group 05 to be placed in the cleared state when the CNC is reset by the RESET key of the MDI unit, the external reset signal ERS<Gn008.7>, the reset and rewind signal RRW<Gn008.6>, or the emergency stop signal.
The table below indicates the correspondence between bits and G code groups
The setting of a bit has the following meaning:
0: Places the G code group in the cleared state.
1: Does not place G code group in the cleared state.

	#7	#6	#5	#4	#3	#2	#1	#0
3409	CFH							

[Input type] Parameter input
[Data type] Bit

#7 CFH When bit 6 (CLR) of parameter No. 3402 is 1, the RESET key on the MDI unit, the external reset signal ERS<Gn008.7>, the reset and rewind signal RRW<Gn008.6>, or emergency stop will,
0: Clear F codes, H codes (for the M series), D codes (for the M series), and T codes (for the T series).
1: Not clear F codes, H codes (for the M series), D codes (for the M series), and T codes (for the T series).

Alarm and message

Number	Message	Description
PS0011	FEED ZERO (COMMAND)	The cutting feedrate instructed by an F code has been set to 0. This alarm is also generated if the F code instructed for the S code is set extremely small in a rigid tapping instruction as the tool cannot cut at the programmed lead.
PS1202	NO F COMMAND AT G93	The F code in inverse time feed mode (G93) is not regarded as being modal. It must be specified for each block.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Cutting feed

7.1.4 Feed per Revolution/Manual Feed per Revolution

Overview

- Feed per revolution

After specifying G95 (G99 for lathe system) (in the feed per revolution mode), the amount of feed of the tool per spindle revolution is to be directly specified by setting a number after F. G95 (G99 for lathe system) is a modal code. Once a G95 is specified, it is valid until G94 (G98 for lathe system) (feed per minute) is specified.
An override of between 0 and 254% (in steps of 1%) can be applied to feed per revolution, using the feedrate override signals *FV0 to *FV7 <Gn012>.

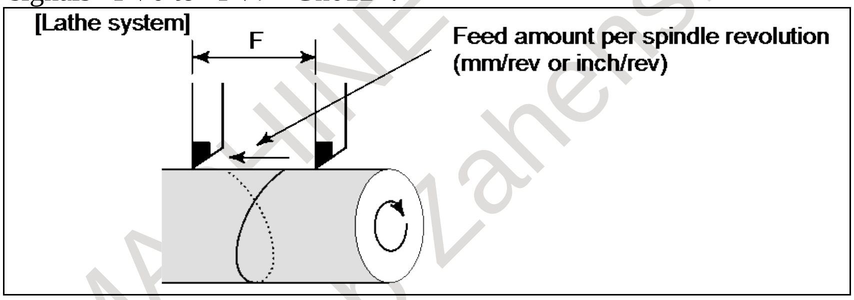


Fig. 7.1.4 (a) Feed per revolution

- Feed per revolution without the position coder

If bit 0 (NPC) of parameter No. 1402 has been set to 1, feed per revolution commands can be specified even when a position coder is not being used. (The CNC converts feed per revolution commands to feed per minute commands.)

- Manual feed per revolution

Jog feedrate can be specified by feed per revolution. An override from 0% to 655.34% (in 0.01% steps) can be applied to manual feed per revolution with the feedrate override signals *JV0 to *JV15 <Gn010,Gn011>.

> **NOTE**
> 1 Manual per revolution feed is not available at the setting of without the position coder (setting bit 0 (NPC) of parameter No.1402 to 1 or bit 1 (FPR) of parameter No.3729 to 1).

NOTE

2 When multi spindle control is enabled and the multiple position coders are used, the target feedback pulses from the position coder for manual per revolution feed is selected with a position coder select signal (PC2SLC<Gn028.7>, PC3SLC<Gn026.0>, PC4SLC <Gn026.1>). And in addition, when multi path control is enabled, an arbitrary position coder can be selected by combining a path spindle feedback selection signals (SLPCA<Gn064.2>, SLPCB<Gn064.3>, SLPCC<Gn403.4>, SLPCD<Gn403.5>). For the selection of a position coder, see the section of "Multi spindle" and "Path spindle control".

3 The command G95 (G99 for lathe system) is not necessary to validate the manual feed per revolution.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1402				JRV				NPC

[Input type] Parameter input
[Data type] Bit path

#0 NPC Feed per revolution without the position coder (function for converting feed per revolution F to feed per minute F in the feed per revolution mode (G95)) is:
0: Not used
1: Used

#4 JRV Jog feed or incremental feed is
0: Performed at feed per minute.
1: Performed at feed per rotation.

NOTE
Specify a feedrate in parameter No. 1423.

1423	Feedrate in manual continuous feed (jog feed) for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
(1) When bit 4 (JRV) of parameter No. 1402 is set to 0 (feed per minute), specify a jog feedrate (feed per minute) under an override of 100%.
(2) When bit 4 (JRV) of parameter No. 1402 is set to 1 (feed per revolution), specify a jog feedrate (feed per revolution) under an override of 100%.

NOTE
This parameter is clamped to the axis-by-axis manual rapid traverse rate (parameter No. 1424).

	#7	#6	#5	#4	#3	#2	#1	#0
3107					GSC			

[Input type] Setting input

[Data type] Bit path

#3 GSC The feedrate to be displayed:
0: Is a feedrate per minute.
1: Follows the setting of bit 5 (FSS) of parameter No. 3191.

	#7	#6	#5	#4	#3	#2	#1	#0
3191			**FSS**					

[Input type] Parameter input
[Data type] Bit path

#5 FSS Feedrate display is:
0: When it meets the following all requirements, display is feedrate per revolution. Besides, display is feedrate per minute.
- Group 05 G-code is feedrate per revolution.
- Other than manual feed, rapid traverse, dry run.
- During axis movement.
1: Fixed to feedrate per revolution.

> **NOTE**
> This parameter is valid only when bit 3 (GSC) of parameter No.3107 is 1.

	#7	#6	#5	#4	#3	#2	#1	#0
3729							**FPRs**	

[Input type] Parameter input
[Data type] Bit spindle

#1 FPRs Feed per revolution (without a position coder) is:
0: Not used for a spindle.
1: Used for a spindle.
In a machine that does not use a position coder, when FPRs is set to 1 for each spindle, feed per revolution can be performed with a spindle command. A feed per revolution is specified with G95 (G99 for lathe systems) in the same way as for normal operation.
When multispindle control is performed, the target spindle for feed per revolution is selected with a position coder select signal (PC2SLC<Gn028.7>, PC3SLC<Gn026.0>, PC4SLC <Gn026.1>).

> **NOTE**
> Enable constant surface speed control (bit 0 (SSC) of parameter No.8133 is 1).

Caution

> ⚠ **CAUTION**
> When the speed of the spindle is low, feedrate fluctuation may occur.
> The slower the spindle rotates, the more frequently feedrate fluctuation occurs.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Cutting feed

7.1.5 One-digit F Code Feed

M

Overview

- One-digit F code feed

When a one-digit number from 1 to 9 is specified after F, the feedrate set for that number in a parameters Nos. 1451 to 1459 is used. When F0 is specified, the rapid traverse rate is applied.
Acceleration/deceleration of the one-digit F code feed is same as cutting feed.
By setting the one-digit F code feed signal F1D <Gn016.7> to 1 and rotating the manual pulse generator, the feedrate of the currently selected number is decreased or increased.
The increment/decrement, ΔF, in feedrate per scale of the manual pulse generator is as follows:

$$\Delta F = \frac{Fmax}{100n}$$

Fmax : Feedrate upper limit for F1-F4 set by parameter 1460, or feedrate upper limit for F5-F9 set by parameter 1461
n : Any value of 1 to 127 set by parameter 1450
The feedrate set or altered is kept even while the power is off. The current feedrate is displayed on the screen.

- Cutting feed override during one-digit F code feed

In case bit 7 (F1O) of parameter No. 1406 is set to 1, the following functions are enabled for a cutting feedrate during one-digit F code feed:

- Feedrate override
- Second feedrate override
- Override cancel

Even during machining using one-digit F code feed, the feedrate can be freely changed, within an applicable override range, from the parameter-set value by changing the override switch.

Signal

One-digit F code feed signal F1D<Gn016.7>

[Classification] Input signal
[Function] Increases or decreases one-digit F speed set by the parameters Nos. 1451 to 1459 using the manual pulse generator.
Since the manual pulse generator may also be used for axis feeding, signal F1D <G016.7> designates which function may be used.
[Operation] When the signal is 1, the one-digit F speed can be increased/decreased using the manual pulse generator.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G016	F1D							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1406								
	F1O							

[Input type] Parameter input
[Data type] Bit path

#7 F1O For the cutting feedrate specified by a single-digit F code (F1 to F9), feedrate override, second feedrate override, and override cancellation are:
0: Disabled.
1: Enabled.

> **NOTE**
> For the F0 feedrate, rapid traverse override is enabled regardless of the setting of this parameter.

1450	
	Change of feedrate for one graduation on the manual pulse generator during one-digit F feed code

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 127

Set the constant that determines the change in feedrate as the manual pulse generator is rotated one graduation during one-digit F feed code.

$$\Delta F = \frac{F\max i}{100n} \quad \text{(where, i=1 or 2)}$$

In the above equation, set n. That is, the number of revolutions of the manual pulse generator, required to reach feedrate Fmaxi is obtained. Fmaxi refers to the upper limit of the feedrate for a one-digit F code feed command, and set it in parameter No. 1460 or 1461.
Fmax1: Upper limit of the feedrate for F1 to F4 (parameter No. 1460)
Fmax2: Upper limit of the feedrate for F5 to F9 (parameter No. 1461)

1451	
	Feedrate for F1
to	to
1459	
	Feedrate for F9

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

These parameters set the feedrates for one-digit F code feed commands F1 to F9. When a one-digit F code feed command is specified, and the feedrate is changed by turning the manual pulse generator, the parameter-set value also changes accordingly.

1460	Upper limit of feedrate for F1 to F4

1461	Upper limit of feedrate for F5 to F9

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the upper limit of feedrate for the one-digit F code feed command.
As the feedrate increases by turning the manual pulse generator, the feedrate is clamped when it reaches the upper limit set. If a one-digit F feed command F1 to F4 is executed, the upper limit is that set in parameter No. 1460. If a one-digit F code feed command F5 to F9 is executed, the upper limit is that set in parameter No. 1461.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Cutting feed

7.1.6 Inverse Time Feed

Overview

Feedrate of the tool can be specified by the move distance of the block and inverse time (FRN).

- Linear interpolation (G01)

FRN=1/Time (min) = Speed/Distance
Speed : mm/ min (metric input)
inch/ min (inch input)
Distance : mm (metric input)
inch (inch input)

- Circular interpolation (G02, G03)

FRN=1/Time (min) = Speed/Circle radius
Speed : mm/ min (metric input)
inch/ min (inch input)
Circle radius : mm (metric input)
inch (inch input)

A value from 0.001 to 9999.999 can be specified as FRN, regardless of whether the input mode is inches or metric, or the increment system is IS-B or IS-C.

F code specification value	FRN
F1	0.001
F1 (*1)	1.000
F1.0	1.000
F9999999	9999.999
F9999 (*1)	9999.000
F9999.999	9999.999

(NOTE)*1: Value specified in fixed-point format with bit 0 (DPI) of parameter No. 3401 set to 1

G code for inverse time feed is a modal G code and belongs to group 05 (includes G code for feed per revolution and G code for feed per minute).
When an F value is specified in inverse time specification mode and the feedrate exceeds the maximum cutting feedrate, the feedrate is clamped to the maximum cutting feedrate.
In the case of circular interpolation, the feedrate is calculated not from the actual amount of movement in the block but from the arc radius. This means that actual machining time is longer when the arc radius is longer than the arc distance and shorter when the arc radius is shorter than the arc distance. Inverse time feed can also be used for cutting feed in a canned cycle.

NOTE

1. In the inverse time specification mode, an F code is not handled as a modal code and therefore needs to be specified in each block. If an F code is not specified, alarm PS1202, "NO F COMMAND AT G93 " is issued.
2. When F0 is specified in inverse time specification mode, alarm PS0011, "FEED ZERO (COMMAND)" is issued.
3. Inverse time feed cannot be used when PMC axis control is in effect.
4. If the calculated cutting feedrate is smaller than the allowable range, alarm PS0011, "FEED ZERO (COMMAND)" is issued.

Example

- For linear interpolation (G01)

$$FRN = \frac{1}{time(min)} = \frac{feedrate}{distance}$$

Feedrate: mm/min (for metric input)
inch/min (for inch input)
Distance: mm (for metric input)
inch (for inch input)

- To end a block in 1 (min)

$$FRN = \frac{1}{time(\text{min})} = \frac{1}{1(\text{min})} = 1$$ Specify F1.0.

- To end a block in 10 (sec)

$$FRN = \frac{1}{time(\text{sec})/60} = \frac{1}{10/60(\text{sec})} = 6$$ Specify F6.0.

- To find the movement time required when F0.5 is specified

$$TIME(\text{min}) = \frac{1}{FRN} = \frac{1}{0.5} = 2$$ 2 (min) is required.

- To find the movement time required when F10.0 is specified

$$TIME(\text{min}) = \frac{1\times 60}{FRN} = \frac{60}{10} = 6$$ 6 (sec) is required.

- For circular interpolation (G02, G03)

$$FRN = \frac{1}{time(\text{min})} = \frac{feedrate}{arcradius}$$

Feedrate: mm/min (for metric input)
inch/min (for inch input)
Arc radius: mm (for metric input)
inch (for inch input)

⚠ **CAUTION**
In circular interpolation, the distance is not an actual distance of the block but the speed is calculated from the circle radius.

Alarm and message

Number	Message	Description
PS0011	FEED ZERO (COMMAND)	The cutting feedrate instructed by an F code has been set to 0. If the calculated cutting feedrate is smaller than the allowable range, this alarm is issued.
PS1202	NO F COMMAND AT G93	The F code in inverse time feed mode (G93) is not regarded as being modal. It must be specified for each block.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Cutting feed

7.1.7 Override

7.1.7.1 Rapid traverse override

Overview

An override of four steps (F0, 25%, 50%, and 100%) can be applied to the rapid traverse rate. F0 is set by a parameter No. 1421.
The 1% step rapid traverse override selection signal HROV<Gn096.7> or 0.1% step rapid traverse override selection signal FHROV<Gn353.7> enables a rapid traverse override to be selected in steps of 1% or 0.1% over a range of 0% to 100%.

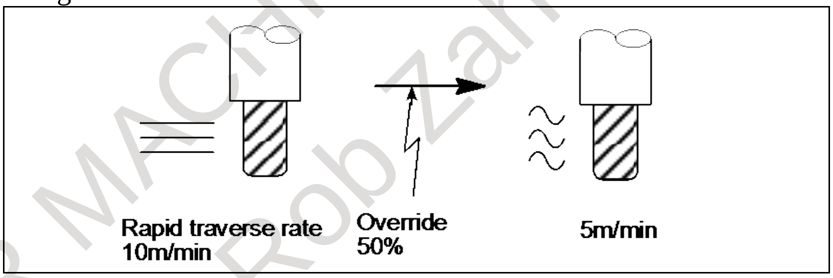


Fig. 7.1.7.1 Rapid traverse override

- Feedrate

Actual feedrate is obtained by multiplying the rapid traverse rate preset by parameter No. 1420 by the override value determined by this signal, whether in automatic or manual operation (including manual reference position return).

- F0 rate

For F0 value, an absolute value is set by parameter No. 1421 within a range of 0 to rapid traverse rate (for each axis).

- 1% step rapid traverse override

1% step rapid traverse override selection signal HROV determines whether rapid traverse override specified with rapid traverse override signals ROV1 and ROV2<Gn014.0,.1> is used or 1% step rapid traverse override is used.
When signal HROV is 0, override is applied to the rapid traverse rate using signals ROV1 and ROV2.
When signal HROV is 1, ROV1 and ROV2 are ignored, 1% step rapid traverse override signals *HROV0 to *HROV6<Gn096.0 to .6> being used to override the rapid traverse rate.

- 0.1% step rapid traverse override

With the 0.1% step rapid traverse override selection signal FHROV, a 1% step rapid traverse override or 0.1% step rapid traverse override can be selected for use.
When the HROV signal is set to 1 and the FHROV signal is set to 0, a 1% step rapid traverse override is applied.
When the HROV and FHROV signals are both set to 1, the 1% step rapid traverse override is disabled, and the rapid traverse override specified by the 0.1% step rapid traverse override signals *FHRO0 to *FHRO9<Gn352.0 to .7,Gn353.0 to .1> is enabled.

- PMC axis control

The rapid traverse override is enabled even in the PMC axis control. The rapid traverse override signals EROV1, EROV2 <Gn150.0,1> or the 1% step rapid traverse override signals *EROV0g to *EROV7g<Gn151> for PMC axis control is enabled even in case that a bit 2 (OVE) of parameter No.8001 is set to 1. The valid signal for PMC axis control follows the setting of bit 1 (OVR) of parameter No. 8013.

- Cutting feedrate override 0%

In case that the RF0 bit (bit 4 of parameter No.1401) is set to 1, when cutting feedrate override is 0% during rapid traverse, the machine tool stops moving.

Signal

Rapid traverse override signal ROV1,ROV2<Gn014.0 to 1>

[Classification] Input signal
[Function] These signals override the rapid traverse rate
[Operation] These two code signals correspond to the rates as follows (Table 7.1.15 (a)):

Table 7.1.7.1 Signals ROV1 to ROV2 and override value

Rapid traverse override		Override value
ROV2	ROV1	
0	0	100%
0	1	50%
1	0	25%
1	1	F0

F0 : Set in parameter No. 1421

1% step rapid traverse override selection signal HROV<Gn096.7>

[Classification] Input signal
[Function] Selects the rapid traverse override signals or the 1% step rapid traverse override signals.
[Operation] When this signal is set to 1, the values of *HROV0 to *HROV6 are enabled. (Rapid traverse override by signals ROV1 and ROV2 is disabled.)
When this signal is set to 0, Rapid traverse override by signals ROV1 and ROV2 is enabled. (the values of *HROV0 to *HROV6 are disabled.)

1% step rapid traverse override signals *HROV0 to *HROV6<Gn096.0 to 6>

[Classification] Input signal
[Function] Applies override to the rapid traverse rate in the range of 0% to 100% in steps of 1%.
[Operation] These seven signals give a binary code indicating an override value applied to the rapid traverse rate.

- When a binary code corresponding to an override value of 101% to 127% is specified, the applied override is clamped at 100%.
- Signals *HROV0 to *HROV6 are inverted signals.

To set an override value of 1%, set signals *HROV0 to *HROV6 to 1111110, which corresponds to a binary code of 0000001.

0.1% step rapid traverse override selection signal FHROV<Gn353.7>

[Classification] Input signal

[Function] Selects whether 1% rapid traverse override signals or 0.1% step rapid traverse override signals are enabled when the HROV signal is set to 1.

[Operation] When this signal is set to 1, the values of *FHRO0 to *FHRO9 are enabled. (The values of the *HROV0 to *HROV6 signals are invalidated.)
When this signal is set to 0, the values of *HROV0 to *HROV6 are enabled. (The values of the *FHRO0 to *FHRO9 signals are invalidated.)

0.1% step rapid traverse override signals *FHRO0 to *FHRO9<Gn352.0 to 7, Gn353.0 to 1>

[Classification] Input signal

[Function] Applies override to the rapid traverse rate in the range of 0% to 100% in steps of 0.1%.

[Operation] These ten signals give a binary code indicating an override value applied to the rapid traverse rate.

- When a binary code corresponding to an override value of 100.1 to 102.3% is specified, the applied override is clamped at 100%.
- Signals *FHRO0 to *FHRO9 are inverted signals.
  To set an override value of 0.1%, set signals *FHRO0 to *FHRO9 to 1111111110, which corresponds to a binary code of 0000000001.

Rapid traverse override signals EROV1, EROV2 <Gn150.0,1>

[Classification] Input signal, path-by-path signal

[Function] Applies rapid traverse override.

[Operation] These signals can be used to select the override for the rapid traverse rate, independently of the CNC, by setting of the bit 2 (OVE) of parameter No. 8001.

Rapid traverse override signals		Override value
EROV2	EROV1	
0	0	100%
0	1	50%
1	0	25%
1	1	F0

F0 is the minimum feedrate specified with parameter No. 1421.

1% step rapid traverse override signals *EROV0g to *EROV7g<Gn151 (, G163, G175, G187)>

[Classification] Input signal, path-by-path signal (group-by-group signal)

[Function] These signals use the same addresses as the feedrate override signals. When bit 1 (OVR) of parameter No. 8013 is set to 1, these signals are used for rapid traverse override.

[Operation] When bit 2 (OVE) of parameter No. 8001 is set to 1 and bit 1 (OVR) of parameter No. 8013 is set to 1, these signals can be used to select the override for the rapid traverse rate, in steps of 1% from 0% to 100%, independently of the CNC. The override is clamped to 100%.
Each corresponding signal has the following significance:

*EROV7 = 100%	*EROV3 = 8%
*EROV6 = 64%	*EROV2 = 4%
*EROV5 = 32%	*EROV1 = 2%
*EROV4 = 16%	*EROV0 = 1%

For example, to set an override of 5%, set signals *EFOV7 to *EFOV0 to 11111010, which corresponds to a binary code of 00000101.

When bit 7 (IFH) of parameter No. 11850 is set to 0, these signals are used as path-by-path signals. When the bit is set to 1, they are used as group-by-group signals.

> **NOTE**
> When all signals are set to “0”, an override of 100% is assumed, while an override of 0% is assumed for *EFOVg.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn014							ROV2	ROV1
Gn096	HROV	*HROV6	*HROV5	*HROV4	*HROV3	*HROV2	*HROV1	*HROV0
Gn150							EROV2	EROV1
Gn151	*EROV7	*EROV6	*EROV5	*EROV4	*EROV3	*EROV2	*EROV1	*EROV0
Gn352	*FHRO7	*FHRO6	*FHRO5	*FHRO4	*FHRO3	*FHRO2	*FHRO1	*FHRO0
Gn353	FHROV						*FHRO9	*FHRO8

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401				RF0				

[Input type] Parameter input
[Data type] Bit path

#4 RF0 When cutting feedrate override is 0% during rapid traverse,
0: The machine tool does not stop moving.
1: The machine tool stops moving.

1421	F0 rate of rapid traverse override for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the F0 rate of the rapid traverse override for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
3002				IOV				

[Input type] Parameter input
[Data type] Bit path

#4 IOV Override-related signal logic is:
0: Used without modification
(A signal of negative logic is used as a negative logic signal, and a signal of positive logic is used as a positive logic signal.)
1: Inverted
(A signal of negative logic is used as a positive logic signal, and a signal of positive logic is used as a negative logic signal.)

The signals indicated below are affected.
Signal of negative logic:
- Feedrate override signals *FV0 to *FV7<Gn012>
- Second feedrate override signals*AFV0 to *AFV7<Gn013>
- 0.01% step second feedrate override signals *APF00 to *APF15<Gn094, Gn095>
- Feedrate override signals (for PMC axis control)
  *EFOV0g to *EFOV7g<G0151/G0163/G0175/G0187>
- Software operator’s panel signals *FV0O to *FV7O<Fn078>

Signals of positive logic:
- Rapid traverse override signals ROV1,ROV2<Gn014.0 to 1>
- Software operator’s panel signals ROV1O,ROV2O<Fn076.4 to 5>
- Rapid traverse override signals (for PMC axis control)
  EROV1g,EROV2g<G0150.0 to 1/G0162.0 to 1/G0174.0 to 1/G0186.0 to 1>

The signals indicated below are not affected.
- 1% step rapid traverse override selection signal HROV <Gn096.7>
- 1% step rapid traverse override signals *HROV0 to *HROV6 <Gn096.0 to 6>
- 0.1% step rapid traverse override selection signal FHROV <Gn353.7>
- 0.1% step rapid traverse override signals *FHRO0 to *FHRO9 <Gn352.0 to 7, Gn353.0 to 1>

	#7	#6	#5	#4	#3	#2	#1	#0
8001						OVE		

[Input type] Parameter input
[Data type] Bit path

#2 OVE Signals related to dry run and override used in PMC axis control
0: Same signals as those used for the CNC
1: Signals specific to the PMC
The signals used depend on the settings of these parameter bits as indicated below.

Signals	Bit 2 (OVE) of parameter No. 8001 = 0 (same signals as those used for the CNC)		Bit 2 (OVE) of parameter No. 8001 = 1 (signals specific to the PMC)	
Feedrate override signals	*FV0 to *FV7	<Gn012>	*EFV0g to *EFV7g	<G0151>
Override cancellation signal	OVC	<Gn006.4>	EOVCg	<G0150.5>
Rapid traverse override signals	ROV1, ROV2	<Gn014.0 to 1>	EROV1g,EROV2g or *EROV0g to *EROV7g	<G0150.0 to 1> <G0151>
Dry run signal	DRN	<Gn046.7>	EDRNg	<G0150.7>
Rapid traverse selection signal	RT	<Gn019.7>	ERTg	<G0150.6>
1% step rapid traverse override selection signal	HROV	<Gn096.7>	None	
1% step rapid traverse override signals	*HROV0 to *HROV6	<Gn096.0 to 6>	None	

Signals	Bit 2 (OVE) of parameter No. 8001 = 0 (same signals as those used for the CNC)		Bit 2 (OVE) of parameter No. 8001 = 1 (signals specific to the PMC)
0.1% step rapid traverse override selection signal	FHROV	<Gn353.7>	None
0.1% step rapid traverse override signals	*FHRO0 to *FHRO9	<Gn352.0 to 7, Gn353.0 to 1>	None

(The signal addresses at PMC selection time are for the first group.)

	#7	#6	#5	#4	#3	#2	#1	#0
8013							**OVR**	

[Input type] Parameter input
[Data type] Bit axis

#1 OVR When bit 2 (OVE) of parameter No. 8001 is set to 1, for rapid traverse override in PMC axis control:

0: Rapid traverse override signals EROV2 and EROV1 <G150.1 and G150.0> for PMC axis control are used.

1: 1% step rapid traverse override signals *EROV7 to *EROV0 <G151> for PMC axis control are used.

(The listed signal addresses when PMC signals are selected are for the 1st group. Actual addresses differs depending on the used group.)

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Rapid traverse override

7.1.7.2 Feedrate override

Overview

A programmed feedrate can be reduced or increased by a percentage (%) selected by the override dial. This feature is used to check a program.
For example, if the programmed feedrate is 100 mm/min, by setting the override value to 50% by using the feedrate override signals *FV0 to *FV7 <Gn012>, it is possible to move the tool at 50 mm/min.

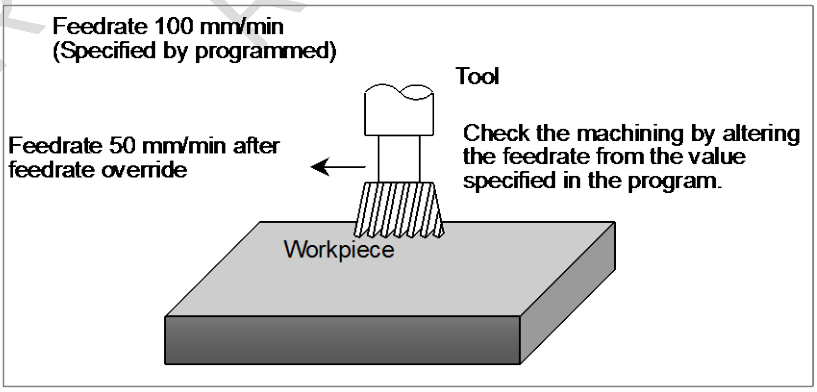


Fig. 7.1.7.2 Feedrate override

Limitation

M

- Cutting feed override during one-digit F code feed

The feedrate override, the second feedrate override and the override cancel are disabled for a cutting feedrate during one-digit F code feed. However, they are enabled in case that a bit 7 (F1O) of parameter No.1406 is set to 1.

- Cutting feed override during rigid tapping

The feedrate override, the second feedrate override and the override cancel are disabled for cutting/extraction operation in rigid tapping. However, they are enabled in case that a bit 4 (OVS) of parameter No.5203 is set to 1.

- Cutting feed override in PMC axis control

The feedrate override and the override cancel are enabled even in the PMC axis control. However, signals specific to PMC are enabled in case that a bit 2 (OVE) of parameter No.8001 is set to 1.

Signal

Feedrate Override signal *FV0 to *FV7<Gn012>

[Classification] Input signal

[Function] These signals override the cutting feedrate. Eight binary code signals correspond to override values as follows:

Override value= $\sum_{i=0}^{7} | 2^{i} \times Vi | \%$

Vi=0 when *FVi is 1 and
Vi=1 when *FVi is 0

These signals have the following weight.

*FV0 : 1% *FV1 : 2%
*FV2 : 4% *FV3 : 8%
*FV4 : 16% *FV5 : 32%
*FV6 : 64% *FV7 : 128 %

When all signals are 0, they are regarded as overriding 0% in the same way as when all signals are 1 .

Thus, the override is selectable in steps over a range of 0 to 254%.

[Operation] Actual feedrate is obtained by multiplying the specified speed by the override value selected by this signal.

The override is regarded as 100%, regardless of this signal, in the following cases:

- Override cancel signal OVC<Gn006.4> is 1.
- During cutting in tap cycle of canned cycle;
- Tapping mode (G63)
- Thread cutting is in progress.

Some examples are listed below.

Table 7.1.7.2 Example of feedrate override

*FV0 to *FV7								Override vlue (%)
7	6	5	4	3	2	1	0	
1	1	1	1	1	1	1	1	0
1	1	1	1	1	1	1	0	1
1	1	1	1	1	1	0	1	2
1	1	1	1	1	1	0	0	3
1	1	1	1	1	0	1	1	4
1	1	1	1	1	0	1	0	5
1	1	1	1	0	1	0	1	10

*FV0 to *FV7								Override vlue (%)
7	6	5	4	3	2	1	0	
1	1	1	1	0	0	0	0	15
1	1	1	0	1	0	1	1	20
1	1	1	0	0	1	1	0	25
1	1	1	0	0	0	0	1	30
1	1	0	1	1	1	0	0	35
1	1	0	1	0	1	1	1	40
1	1	0	1	0	0	1	0	45
1	1	0	0	1	1	0	1	50
1	1	0	0	0	0	1	1	60
1	0	1	1	1	0	0	1	70
1	0	1	0	1	1	1	1	80
1	0	1	0	0	1	0	1	90
1	0	0	1	1	0	1	1	100
0	1	1	0	1	0	0	1	150
0	0	1	1	0	1	1	1	200
0	0	0	0	0	0	0	1	254
0	0	0	0	0	0	0	0	0

The value is calculated as follows.

1. In case that the override is 2%,
   (1) Convert to binary data. 0000 0010
   (2) Do logical NOT of binary data. 1111 1101
2. In case that the input signal is “1110 1110”,
   (1) Do logical NOT of binary data. 0001 0001
   (2) Convert to decimal data. 17%

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn012	*FV7	*FV6	*FV5	*FV4	*FV3	*FV2	*FV1	*FV0

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401				RF0				

[Input type] Parameter input
[Data type] Bit path

#4 RF0 When cutting feedrate override is 0% during rapid traverse,
0: The machine tool does not stop moving.
1: The machine tool stops moving.

	#7	#6	#5	#4	#3	#2	#1	#0
1406								
	F1O							

[Input type] Parameter input
[Data type] Bit path

#7 F1O For the cutting feedrate specified by a single-digit F code (F1 to F9), feedrate override, second feedrate override, and override cancellation are:
0: Disabled.
1: Enabled.

> **NOTE**
> For the F0 feedrate, rapid traverse override is enabled regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3002				IOV				

[Input type] Parameter input
[Data type] Bit path

#4 IOV Override-related signal logic is:
0: Used without modification
(A signal of negative logic is used as a negative logic signal, and a signal of positive logic is used as a positive logic signal.)
1: Inverted
(A signal of negative logic is used as a positive logic signal, and a signal of positive logic is used as a negative logic signal.)

The signals indicated below are affected.
Signal of negative logic:
- Feedrate override signals *FV0 to *FV7<Gn012>
- Second feedrate override signals*AFV0 to *AFV7<Gn013>
- 0.01% step second feedrate override signals *APF00 to *APF15<Gn094, Gn095>
- Feedrate override signals (for PMC axis control)
  *EFOV0g to *EFOV7g<G0151/G0163/G0175/G0187>
- Software operator's panel signals *FV0O to *FV7O<Fn078>

Signals of positive logic:
- Rapid traverse override signals ROV1,ROV2<Gn014.0, Gn014.1>
- Software operator's panel signals ROV1O,ROV2O<Fn076.4 to 5>
- Rapid traverse override signals (for PMC axis control)
  EROV1g,EROV2g<G0150.0 to 1/G0162.0 to 1/G0174.0 to 1/G0186.0 to 1>

The signals indicated below are not affected.
- 1% step rapid traverse override selection signal HROV <Gn096.7>
- 1% step rapid traverse override signals *HROV0 to *HROV6 <Gn096.0 to 6>
- 0.1% step rapid traverse override selection signal FHROV <Gn353.7>
- 0.1% step rapid traverse override signals *FHRO0 to *FHRO9 <Gn352.0 to 7, Gn353.0 to 1>

	#7	#6	#5	#4	#3	#2	#1	#0
5203				OVS				

[Input type] Parameter input
[Data type] Bit path

#4 OVS In rigid tapping, override by the feedrate override signals *FV0 to *FV7 <Gn012> and the second feedrate override signals *AFV0 to *AFV7 <Gn013> (or 0.01% step second feedrate override signals *APF00 to *APF15 <Gn094, Gn095>), and the cancellation of override by the override cancel signal OVC <Gn006.4> is:
0: Disabled.
1: Enabled.
When feedrate override is enabled, extraction override is disabled.
The spindle override is clamped to 100% during rigid tapping, regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
8001						OVE		

[Input type] Parameter input
[Data type] Bit path

#2 OVE Signals related to dry run and override used in PMC axis control
0: Same signals as those used for the CNC
1: Signals specific to the PMC

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Cutting feed

7.1.7.3 Second feedrate override

Overview

An override can be applied to the cutting feedrate. The override is applied to the feedrate to which an override has been applied with the first feedrate override signals *FV0 to *FV7 <Gn012>.

(Actual feedrate) = F × FV_1 × FV_2

F: Specified cutting feedrate
FV_1: Feedrate override (%)
FV_2: Second feedrate override (%)

A percentage from 0 to 254 % can be selected in steps of 1% as the second feedrate override.
In case that a bit 3 (OV2) of parameter No.1402 is set to 1, a percentage from 0 to 655.34% can be selected in steps 0.01% as the second feedrate override.

Limitation

M

- Cutting feed override during one-digit F code feed

The feedrate override, the second feedrate override and the override cancel are disabled for a cutting feedrate during one-digit F code feed. However, they are enabled in case that a bit 7 (F1O) of parameter No.1406 is set to 1.

- Cutting feed override during rigid tapping

The feedrate override, the second feedrate override and the override cancel are disabled for cutting/extraction operation in rigid tapping. However, they are enabled in case that a bit 4 (OVS) of parameter No.5203 is set to 1.

- Cutting feed override in PMC axis control

The second feedrate override is disabled in the PMC axis control.

Signal

Second feedrate override signals *AFV0 to *AFV7<Gn013>

[Classification] Input signal
[Function] The cutting feedrate is multiplied by the second feedrate override. These eight binary code signals correspond to the override values as follows.

$$\text{Override value}=\sum_{i=0}^{7}|2^{i}\times Vi|\,\%$$

Vi=0 when *AFVi is 1 and

Vi=1 when *AFVi is 0
These signals have the following weight.
*AFV0 : 1%   *AFV1 : 2%
*AFV2 : 4%   *AFV3 : 8%
*AFV4 : 16%   *AFV5 : 32%
*AFV6 : 64%   *AFV7 : 128 %
If all signals are 0 or 1, the override is regarded as 0%. The override is selectable in steps of 1% over a range of 0 to 254%.

[Operation] In case that a bit 3 (OV2) of parameter No.1402 is set to 0, those signals are enabled.
In cutting feed during automatic operation, the actual feedrate is obtained as follows.
Actual feedrate = (Specified speed) × (First override) × (Override value specified by this signal)
Since the condition under which the first override is ignored and is set to 100% remains valid, the second feedrate override is also assumed to 100%.
Some examples are listed below.

Table 7.1.7.3 (a) Example of second feedrate override

*AFV0 to *AFV7								Override vlue (%)
7	**6**	**5**	**4**	**3**	**2**	**1**	**0**	
1	1	1	1	1	1	1	1	0
1	1	1	1	1	1	1	0	1
1	1	1	1	1	1	0	1	2
1	1	1	1	1	1	0	0	3
1	1	1	1	1	0	1	1	4
1	1	1	1	1	0	1	0	5
1	1	1	1	0	1	0	1	10
1	1	1	1	0	0	0	0	15
1	1	1	0	1	0	1	1	20
1	1	1	0	0	1	1	0	25
1	1	1	0	0	0	0	1	30
1	1	0	1	1	1	0	0	35
1	1	0	1	0	1	1	1	40
1	1	0	1	0	0	1	0	45
1	1	0	0	1	1	0	1	50
1	1	0	0	0	0	1	1	60
1	0	1	1	1	0	0	1	70
1	0	1	0	1	1	1	1	80
1	0	1	0	0	1	0	1	90
1	0	0	1	1	0	1	1	100
0	1	1	0	1	0	0	1	150
0	0	1	1	0	1	1	1	200
0	0	0	0	0	0	0	1	254
0	0	0	0	0	0	0	0	0

The value is calculated as follows.

1. In case that the override is 2%,
   (1) Convert to binary data.   0000 0010
   (2) Do logical NOT of binary data.   1111 1101
2. In case that the input signal is “1110 1110”,
   (1) Do logical NOT of binary data.   0001 0001
   (2) Convert to decimal data.   17%

> **NOTE**
> In case that the second feedrate override is enabled and the second feedrate override is not used, those signals must be set so that the override value is set to 100%.

0.01% step second feedrate override signals *APF00 to *APF15<Gn094, Gn095>

[Classification] Input signal

[Function] The cutting feedrate is multiplied by the second feedrate override. These sixteen binary code signals correspond to the override values as follows.

$$\text{Override value}=\sum_{i=0}^{15}|\,2^{i}\times Vi\,|\times 0.01\%$$

Vi=0 when *APFi is 1 and
Vi=1 when *APFi is 0
These signals have the following weight.
*APF00=0.01%, *APF01=0.02%, *APF02=0.04%, *APF03=0.08%
*APF04=0.16%, *APF05=0.32%, *APF06=0.64%, *APF07=1.28%
*APF08=2.56%, *APF09=5.12%, *APF10=10.24%, *APF11=20.48%
*APF12=40.96%, *APF13=81.92%, *APF14=163.84%, *APF15=327.68%
If all signals are 0 or 1, the override is regarded as 0%. The override is selectable in steps of 0.01% over a range of 0 to 655.34%.

[Operation] In case that a bit 3 (OV2) of parameter No.1402 is set to 1, those signals are enabled.
In cutting feed during automatic operation, the actual feedrate is obtained as follows.
Actual feedrate = (Specified speed) × (First override) × (Override value specified by this signal)
Since the condition under which the first override is ignored and is set to 100% remains valid, the second feedrate override is also assumed to 100%.
Some examples are listed below.

Table 7.1.7.3 (b) Example of 0.01% step second feedrate override

*APF00～*APF15																Override value (%)
15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0	
1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	0.00
1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	0	0.01
1	1	1	1	1	1	1	1	1	1	1	1	1	1	0	1	0.02
1	1	1	1	1	1	1	1	1	1	1	1	1	0	1	0	0.05
1	1	1	1	1	1	1	1	1	1	1	1	0	1	0	1	0.10
1	1	1	1	1	1	1	1	1	1	1	0	1	0	1	1	0.20
1	1	1	1	1	1	1	1	1	1	0	0	1	1	0	1	0.50
1	1	1	1	1	1	1	1	1	0	0	1	1	0	1	1	1.00
1	1	1	1	1	1	1	1	0	0	1	1	0	1	1	1	2.00
1	1	1	1	1	1	1	0	0	0	0	0	1	0	1	1	5.00
1	1	1	1	1	1	0	0	0	0	0	1	0	1	1	1	10.00
1	1	1	1	1	0	0	0	0	0	1	0	1	1	1	1	20.00
1	1	1	0	1	1	0	0	0	1	1	1	0	1	1	1	50.00
1	1	0	1	1	0	0	0	1	1	1	0	1	1	1	1	100.00
1	0	1	1	0	0	0	1	1	1	0	1	1	1	1	1	200.00
0	0	1	1	1	1	0	0	1	0	1	0	1	1	1	1	500.00
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1	655.34
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0.00

The value is calculated as follows.

1. In case that the override is 23.45%,
   (1) Multiply by 100 — 2345
   (2) Convert to binary data. — 0000 1001 0010 1001

(3) Do logical NOT of binary data. 1111 0110 1101 0110

2. In case that the input signal is “1110 1110 1110 1110”,
   (1) Do logical NOT of binary data. 0001 0001 0001 0001
   (2) Convert to decimal data. 4369
   (3) Multiply by 0.01 43.69%

> **NOTE**
> In case that the second feedrate override is enabled and the second feedrate override is not used, those signals must be set so that the override value is set to 100%.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn013	*AFV7	*AFV6	*AFV5	*AFV4	*AFV3	*AFV2	*AFV1	*AFV0
Gn094	*APF07	*APF06	*APF05	*APF04	*APF03	*APF02	*APF01	*APF00
Gn095	*APF15	*APF14	*APF13	*APF12	*APF11	*APF10	*APF09	*APF08

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1402					OV2			

[Input type] Parameter input
[Data type] Bit path

#3 OV2 Signals used for 2nd feedrate override are
0: *AFV0 to *AFV7 <Gn013> (specified every 1%)
1: *APF00 to *APF15 <Gn094, Gn095> (specified every 0.01%)

	#7	#6	#5	#4	#3	#2	#1	#0
1406								
	F1O							

[Input type] Parameter input
[Data type] Bit path

#7 F1O For the cutting feedrate specified by a single-digit F code (F1 to F9), feedrate override, second feedrate override, and override cancellation are:
0: Disabled.
1: Enabled.

> **NOTE**
> For the F0 feedrate, rapid traverse override is enabled regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3002				IOV				

[Input type] Parameter input
[Data type] Bit path

#4 IOV Override-related signal logic is:

0: Used without modification
(A signal of negative logic is used as a negative logic signal, and a signal of positive logic is used as a positive logic signal.)

1: Inverted
(A signal of negative logic is used as a positive logic signal, and a signal of positive logic is used as a negative logic signal.)

The signals indicated below are affected.

Signal of negative logic:
- Feedrate override signals *FV0 to *FV7<Gn012>
- Second feedrate override signals*AFV0 to *AFV7<Gn013>
- 0.01% step second feedrate override signals *APF00 to *APF15<Gn094, Gn095>
- Feedrate override signals (for PMC axis control)
  *EFOV0g to *EFOV7g<G0151/G0163/G0175/G0187>
- Software operator’s panel signals *FV0O to *FV7O<Fn078>

Signals of positive logic:
- Rapid traverse override signals ROV1,ROV2<Gn014.0 to 1>
- Software operator’s panel signals ROV1O,ROV2O<Fn076.4 to 5>
- Rapid traverse override signals (for PMC axis control)
  EROV1g,EROV2g<G0150.0 to 1/G0162.0 to 1/G0174.0 to 1/G0186.0 to 1>

The signals indicated below are not affected.
- 1% step rapid traverse override selection signal HROV <Gn096.7>
- 1% step rapid traverse override signals *HROV0 to *HROV6 <Gn096.0 to 6>
- 0.1% step rapid traverse override selection signal FHROV <Gn353.7>
- 0.1% step rapid traverse override signals *FHRO0 to *FHRO9 <Gn352.0 to 7, Gn353.0 to 1>

	#7	#6	#5	#4	#3	#2	#1	#0
5203				OVS				

[Input type] Parameter input
[Data type] Bit path

#4 OVS In rigid tapping, override by the feedrate override signals *FV0 to *FV7 <Gn012> and the second feedrate override signals *AFV0 to *AFV7 <Gn013> (or 0.01% step second feedrate override signals *APF00 to *APF15 <Gn094, Gn095>), and the cancellation of override by the override cancel signal OVC <Gn006.4> is:

0: Disabled.

1: Enabled.

When feedrate override is enabled, extraction override is disabled.
The spindle override is clamped to 100% during rigid tapping, regardless of the setting of this parameter.

7.1.7.4 Override cancel

Overview

The override cancel signal OVC<Gn006.4> fixes the feedrate override to 100%.

Limitation

M

- Cutting feed override during one-digit F code feed

The feedrate override, the second feedrate override and the override cancel are disabled for a cutting feedrate during one-digit F code feed. However, they are enabled in case that a bit 7 (F1O) of parameter No.1406 is set to 1.

- Cutting feed override during rigid tapping

The feedrate override, the second feedrate override and the override cancel are disabled for cutting/extraction operation in rigid tapping. However, they are enabled in case that a bit 4 (OVS) of parameter No.5203 is set to 1.

- Cutting feed override in PMC axis control

The feedrate override and the override cancel are enabled even in the PMC axis control. However, signals specific to PMC are enabled in case that a bit 2 (OVE) of parameter No.8001 is set to 1.

Signal

Override cancel signal OVC<Gn006.4>

[Classification] Input signal

[Function] Feedrate override is fixed to 100%.

[Operation] When the signal is 1, the CNC operates as follows:

- The feedrate override is fixed to 100% regardless of the feedrate override signals *FV0 to *FV7 <Gn012> or second feedrate override signals *AFV0 to *AFV7 <Gn013> (or 0.01% step second feedrate override signals *APF00 to *APF15 <Gn094, Gn095>). That is, the specified feedrate is used as is.
- Rapid traverse override and spindle speed override are not affected.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn006				OVC				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1406								
	F1O							

[Input type] Parameter input

[Data type] Bit path

#7 F1O For the cutting feedrate specified by a single-digit F code (F1 to F9), feedrate override, second feedrate override, and override cancellation are:

0: Disabled.

1: Enabled.

NOTE
For the F0 feedrate, rapid traverse override is enabled regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5203				OVS				

[Input type] Parameter input
[Data type] Bit path

#4 OVS In rigid tapping, override by the feedrate override signals *FV0 to *FV7 <Gn012> and the second feedrate override signals *AFV0 to *AFV7 <Gn013> (or 0.01% step second feedrate override signals *APF00 to *APF15 <Gn094, Gn095>), and the cancellation of override by the override cancel signal OVC <Gn006.4> is:
0: Disabled.
1: Enabled.
When feedrate override is enabled, extraction override is disabled.
The spindle override is clamped to 100% during rigid tapping, regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
8001						OVE		

[Input type] Parameter input
[Data type] Bit path

#2 OVE Signals related to dry run and override used in PMC axis control
0: Same signals as those used for the CNC
1: Signals specific to the PMC

7.1.8 Automatic Corner Override

M

The movement of a tool in an inner corner or an inner arc is automatically decelerated during cutter compensation, so that the load on the tool is reduced, and a surface obtained by cutting becomes smooth.

7.1.8.1 Inner corner automatic override (G62)

Explanation

- Overriding conditions

When G62 is specified, and the tool path with cutter compensation applied forms an inner corner, the feedrate is automatically overridden at both ends of the corner.

There are four types of inner corners (Fig. 7.1.8.1 (a)).

$2° \leq \theta \leq \theta p \leq 178°$ in Fig. 7.1.8.1 (a)

θp is a value set with parameter No. 1711. When θ is approximately equal to θp, the inner corner is determined with an error of 0.001° or less.

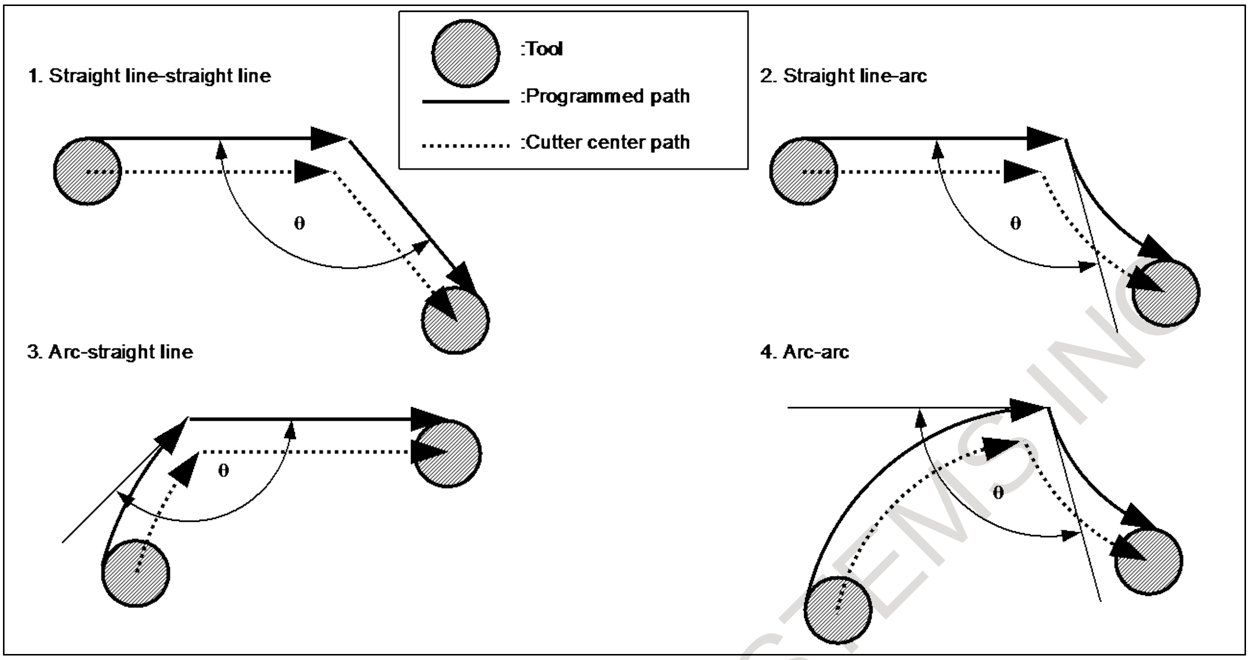


Fig. 7.1.8.1 (a) Inner corner

- Overriding range

When an inner corner is found, the feedrate is overridden before and after the intersection of the corresponding tool paths. Feedrate override distances Le and Ls are linear dimensions between the intersection of the tool paths and a point on either cutter center path (Fig. 7.1.8.1 (b), Fig. 7.1.8.1 (c), Fig. 7.1.8.1 (d)). Le and Ls are specified in parameters Nos.1713 and 1714.

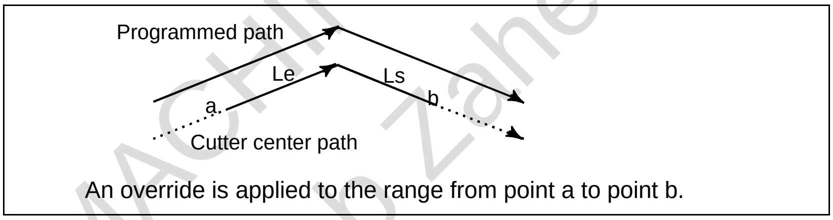


Fig. 7.1.8.1 (b) Overriding range (straight line - straight line)

An arc-shaped path is overridden if the start point and end point of the arc is in the same quadrant or in two adjacent quadrants (Fig. 7.1.8.1 (c)).

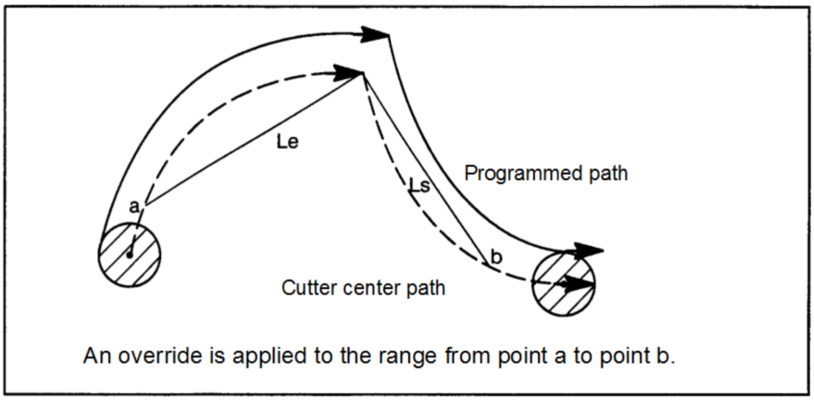


Fig. 7.1.8.1 (c) Overriding range (arc - arc)

On arc-shaped path <2> programmed as indicated below, an override occurs between points a and b and between points c and d (Fig. 7.1.8.1 (d)).

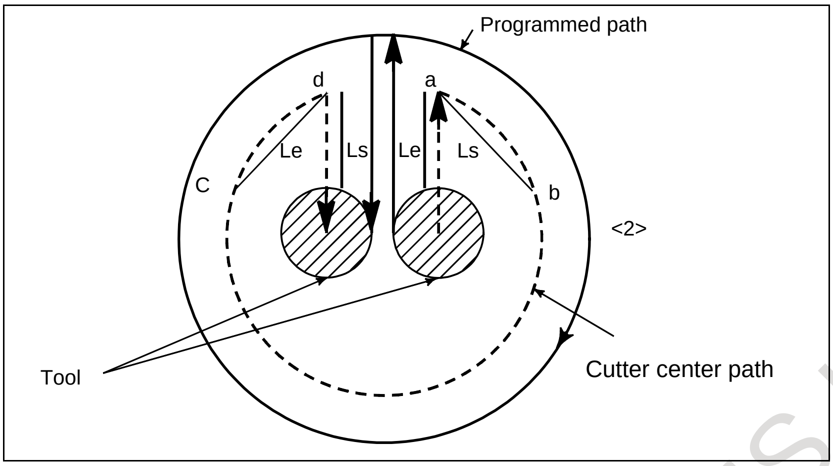


Fig. 7.1.8.1 (d) Overriding range (straight line - arc, arc - straight line)

- Override value

An override value is set with parameter No. 1712. An override value is valid even for dry run and F1-digit feed specification.
In the feed per minute mode, the actual feedrate is as follows:
F × (inner corner automatic override) × (feedrate override)

Limitation

- Look-ahead acceleration/deceleration before interpolation

The inner corner override function is disabled during look-ahead acceleration/deceleration before interpolation.

- Start-up, G41, G42

The inner corner override function is not carried out for a corner preceded by a cutter compensation start-up block or a corner followed by a block containing G41 or G42,.

- Offset value

If the offset value is zero, the inner corner override function is not executed.

7.1.8.2 Internal circular cutting feedrate change

For internally offset circular cutting, the feedrate on a programmed path is set to a specified feedrate (F) by specifying the circular cutting feedrate with respect to F, as indicated below (Fig. 7.1.8.2). This function is valid in the cutter compensation mode, regardless of the G62 code.

$$F \times \frac{Rc}{Rp}$$

Rc: Cutter center path radius
Rp: Programmed tool path radius

It is also valid for the dry run and the one digit F cord feed.

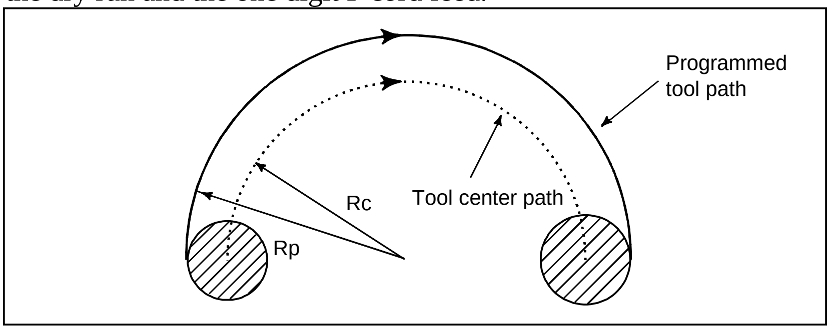


Fig. 7.1.8.2 Internal circular cutting feedrate change

If Rc is much smaller than Rp, Rc/Rp≒0; the tool stops. A minimum deceleration ratio (MDR) is to be specified with parameter No. 1710. When Rc/Rp≤MDR, the feedrate of the tool is (F×MDR).
If parameter No. 1710 is 0, the minimum deceleration ratio (MDR) is 100%.

> ⚠ **CAUTION**
> When internal circular cutting must be performed together with automatic override for inner corners, the feedrate of the tool is as follows:
> F×Rc/Rp×(inner corner override)×(feedrate override)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1602						**CAF**		

[Input type] Parameter input
[Data type] Bit path

#2 CAF In cutter compensation mode or tool nose radius compensation mode, the circular cutting feedrate change is;
0: Performed on the inner arc only.
1: Performed on the inner and outer arcs.

1710	Minimum deceleration ratio (MDR) for inner circular cutting feedrate change by automatic corner override

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 100

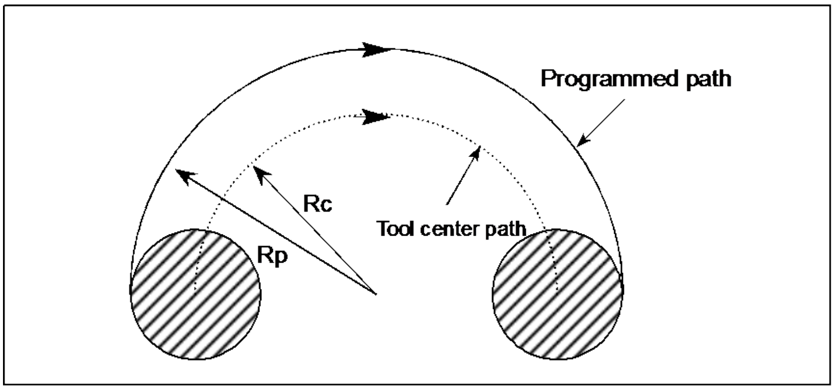
Set a minimum deceleration ratio (MDR) for an inner circular cutting feedrate change by automatic corner override.

In the case of circular cutting offset inward, the actual feedrate is determined by a specified feedrate (F) as follows:

$$F \times \frac{Rc}{Rp}$$

Rc: Radius of tool center path
Rp: Programmed radius

Thus, the feedrate along the programmed path satisfies the specified value of F.



However, if Rc is too small when compared with Rp, Rc/Rp ≒ 0 results to stop the tool. So, a minimum deceleration ratio (MDR) is set, and the feedrate of the tool is set to F×(MDR) when Rc/Rp ≤ MDR.
When this parameter is 0, the minimum deceleration ratio (MDR) is 100%.

1711	Inner determination angle (θp) for inner corner override

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 2 to 178

Set an inner determination angle for inner corner override in automatic corner overriding.

1712	Override value for inner corner override

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 1 to 100

Set an inner corner override value in automatic corner overriding.

1713	Start distance (Le) for inner corner override

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

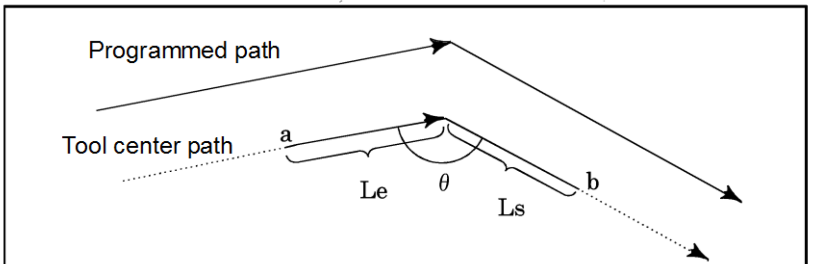
Set a start distance for inner corner override in automatic corner overriding.

1714	End distance (Ls) for inner corner override

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set an end distance for inner corner override in automatic corner overriding.

When $\theta \leq \theta p$, an inner corner is assumed. (Parameter No. 1711 is used to set θp.)
When a corner is determined to be an inner corner, an override is applied to the feedrate in the range of Le in the previous block from the intersection of the corner and in the range of Ls in the next block from the intersection of the corner.
Distances Le and Ls represent linear distances from the intersection of a corner to points on the tool center path.
Le and Ls are set in parameters Nos. 1713 and 1714.



An override is applied to the range from point a to point b.

7.1.9 External Deceleration

Overview

The control axis is externally decelerated. The feedrate is decelerated by the external deceleration signals from the machine. The deceleration rate is set by the parameters.
The external deceleration signal are provided for each axis and direction.
As five types of deceleration condition settings can be dynamically selected by the signals.
External deceleration can be applied to rapid traverse rate, cutting feedrate, and manual handle feedrate.
For manual handle federate, external deceleration can be applied under the following conditions.

- Rapid traverse and cutting feed

Five types of deceleration conditions can be set by the parameters.
When deceleration conditions specified by multiple external signals input during machining are conflicted, the condition with the lowest external deceleration rate is applied.
External deceleration settings 2, 3. 4, and 5 can be switched between enabled and disabled by the parameters.

- Manual handle feed

Five types of deceleration conditions can be set by the parameters. For handle feed, the maximum feedrate is switched when the external deceleration signal in the positive or negative direction for the handle axis is set to 0. When multiple conditions are conflicted, the condition with the lowest maximum feedrate is applied.

- *Setting the parameters*

Valid / invalid of the external deceleration signals in the cutting feed and the external deceleration rate for linear interpolation type positioning is set by the following parameters.

No.1005#4 In cutting feed, an external deceleration signal in the + direction for each axis is disable(0) / enable(1)

No.1005#5 In cutting feed, an external deceleration signal in the - direction for each axis is disable(0) / enable(1)

No.1405#5 As the external deceleration rate for linear interpolation type positioning, the external deceleration rate for cutting feed is used(0) / the external deceleration rate in rapid traverse for the 1st axis in the entire system is used(1)

External deceleration settings 1～5 are set by the following parameters.

External deceleration / Meaning	Setting 1	Setting 2	Setting 3	Setting 4	Setting 5
Valid / invalid of the external deceleration setting	-	No.1406#0	No.1406#1	No.12750#0	No.12750#1
External deceleration rate of cutting feed	No.1426	No.1440	No.1443	No.12751	No.12754
External deceleration rate of rapid traverse for each axis	No.1427	No.1441	No.1444	No.12752	No.12755
Maximum manual handle feedrate for each axis	No.1434	No.1442	No.1445	No.12753	No.12756

- Manual continuous feed

When the manual handle feed is enabled (bit 0 (HPG) of parameter No.8131) is 1), and the parameter No.7100#0 is set to 1, the external deceleration is enabled as same condition for the manual handle feed. Besides, the external deceleration is enabled as same condition for the rapid feed.

Signal

External deceleration signals 1
*+ED1 to *+ED8<Gn118>,*-ED1 to *-ED8<Gn120>
External deceleration signals 2
*+ED21 to *+ED28<Gn101>,*-ED21 to *-ED28<Gn103>
External deceleration signals 3
*+ED31 to *+ED38<Gn107>,*-ED31 to *-ED38<Gn109>
External deceleration signals 4
*+ED41 to *+ED48<Gn341>,*-ED41 to *-ED48<Gn342>
External deceleration signals 5
*+ED51 to *+ED58<Gn343>,*-ED51 to *-ED58<Gn344>

[Classification] Input signal

[Function] These signals specifies whether to apply external deceleration to each direction for each control axis. The sign + or - preceding ED indicates the direction of feed which is decelerated. The number following ED indicates the number of external deceleration setting. The last number indicates the number of a control axis.

* xEDyz

z : 1 ….. The 1st axis is decelerated.
2 ….. The 2nd axis is decelerated.
3 ….. The 3rd axis is decelerated.
:

y : None.. External deceleration setting 1 is selected.
2 ….. External deceleration setting 2 is selected.
3 ….. External deceleration setting 3 is selected.
4 ….. External deceleration setting 4 is selected.
5 ….. External deceleration setting 5 is selected.

x : + : The feed is decelerated in the plus (+) direction.
- : The feed is decelerated in the minus (-) direction.

[Operation] When a signal becomes 0, the corresponding axis decelerates in the specified direction.
If a specified feedrate is lower than the external deceleration rate, the specified feedrate is selected.

Manual handle feed maximum feedrate change signal HNDLF<Gn023.3>

[Classification] Input signal

[Function] This signal switches for the maximum handle feedrate.

[Operation] When this signal is 0, clamping is performed by the manual rapid traverse rate (parameter No. 1424) or parameter No. 1420. The external deceleration function is disabled for handle feed.
When this signal is 1, clamping is performed by the maximum handle feedrate (parameter No. 1434). The external deceleration function is enabled for handle feed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn118	*+ED8	*+ED7	*+ED6	*+ED5	*+ED4	*+ED3	*+ED2	*+ED1
Gn120	*-ED8	*-ED7	*-ED6	*-ED5	*-ED4	*-ED3	*-ED2	*-ED1
Gn101	*+ED28	*+ED27	*+ED26	*+ED25	*+ED24	*+ED23	*+ED22	*+ED21

Gn103	*-ED28	*-ED27	*-ED26	*-ED25	*-ED24	*-ED23	*-ED22	*-ED21
Gn107	*+ED38	*+ED37	*+ED36	*+ED35	*+ED34	*+ED33	*+ED32	*+ED31
Gn109	*-ED38	*-ED37	*-ED36	*-ED35	*-ED34	*-ED33	*-ED32	*-ED31
Gn341	*+ED48	*+ED47	*+ED46	*+ED45	*+ED44	*+ED43	*+ED42	*+ED41
Gn342	*-ED48	*-ED47	*-ED46	*-ED45	*-ED44	*-ED43	*-ED42	*-ED41
Gn343	*+ED58	*+ED57	*+ED56	*+ED55	*+ED54	*+ED53	*+ED52	*+ED51
Gn344	*-ED58	*-ED57	*-ED56	*-ED55	*-ED54	*-ED53	*-ED52	*-ED51
Gn023					HNDLF			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1005			EDMx	EDPx				

[Input type] Parameter input
[Data type] Bit axis

#4 EDPx In cutting feed, an external deceleration signal in the + direction for each axis is:
0: Invalid
1: Valid

#5 EDMx In cutting feed, an external deceleration signal in the - direction for each axis is:
0: Invalid
1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
1405			EDR					

[Input type] Parameter input
[Data type] Bit path

#5 EDR As the external deceleration rate for linear interpolation type positioning:
0: The external deceleration rate for cutting feed is used.
1: The external deceleration rate in rapid traverse for the 1st axis in the entire system is used.

[Example of external deceleration 1]
When this parameter is set to 0, the value of parameter No. 1426 is used as the external deceleration rate for external deceleration 1.
When this parameter is set to 1, the parameter No. 1427 for the 1st axis in the entire system is used as the external deceleration rate for external deceleration 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1406							EX3	EX2

[Input type] Parameter input
[Data type] Bit path

#0 EX2 External deceleration function setting 2 is:
- 0: Invalid
- 1: Valid

#1 EX3 External deceleration function setting 3 is:
- 0: Invalid
- 1: Valid

1426	External deceleration rate of cutting feed

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set an external deceleration rate for cutting feed or positioning of linear interpolation type (G00).

1427	External deceleration rate of rapid traverse for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the external deceleration rate of rapid traverse for each axis.

1434	Maximum manual handle feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set a maximum manual handle feedrate for each axis in case of manual handle feedrate switch signal HNDLF<Gn023.3>=1.

1440	External deceleration rate setting 2 in cutting feed

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set external deceleration rate 2 for cutting feed or positioning of linear interpolation type (G00).

1441	External deceleration rate setting 2 for each axis in rapid traverse

[Input type] Parameter input
[Data type] Real axis

[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set external deceleration rate 2 for each axis in rapid traverse.

1442	Maximum manual handle feedrate setting 2 for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set a maximum manual handle feedrate 2 for each axis.

1443	External deceleration rate setting 3 in cutting feed

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set external deceleration rate 3 for cutting feed or positioning of linear interpolation type (G00).

1444	External deceleration rate setting 3 for each axis in rapid traverse

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set external deceleration rate 3 for each axis in rapid traverse.

1445	Maximum manual handle feedrate setting 3 for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set a maximum manual handle feedrate 3 for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
12750							EX5	EX4

[Input type] Parameter input
[Data type] Bit path

#0 EX4 External deceleration function setting 4 is:
0: Disabled.
1: Enabled.

#1 EX5 External deceleration function setting 5 is:
0: Disabled.
1: Enabled.

12751	External deceleration rate setting 4 in cutting feed

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set external deceleration rate 4 for cutting feed or positioning of linear interpolation type (G00).

12752	External deceleration rate setting 4 for each axis in rapid traverse

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set external deceleration rate 4 for each axis in rapid traverse.

12753	Maximum manual handle feedrate setting 4 for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set a maximum manual handle feedrate 4 for each axis.

12754	External deceleration rate setting 5 in cutting feed

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set external deceleration rate 5 for cutting feed or positioning of linear interpolation type (G00).

12755	External deceleration rate setting 5 for each axis in rapid traverse

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)

[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set external deceleration rate 5 for each axis in rapid traverse.

12756	Maximum manual handle feedrate setting 5 for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set a maximum manual handle feedrate 5 for each axis.

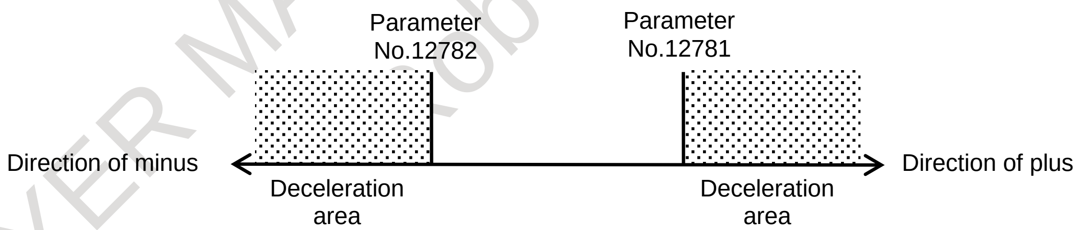
7.1.9.1 Deceleration area setting

When the axis is moved to outside specified area, the feedrate can be automatically decelerated.
In area that the movement condition is severe (the end point of stroke ,etc), the feedrate can be decreased.
Therefore, the feedrate ban be increased in other areas.
This function is activated when bit 0 (ADE) of parameter No.12780 is set to 1.

NOTE
1 For this function, enable "External Deceleration"(bit 2(EDC) of parameter No.8131 is 1).
2 This function is also invalid for functions in which external deceleration is invalid.

Explanation

Set to the deceleration area by the parameter No.12781 and No.12782.
If the axis is outside this area, the feedrate is automatically reduced so that the feedrate after applying the override is less than the speed set by parameter No. 12783(rapid traverse) or No.12784(cutting feed).



	Parameter	
	Rapid traverse	Cutting feed
Deceleration rate	12783	12784

3-dimensional coordinate conversion / Tilted working plane command

When this function is used in 3-dimensional coordinate conversion / tilted working plane indexing, set the linear interpolation positioning (parameter LRP (No.1401#1)=1).

- Use with other functions

This function is disabled when the following functions are used together.
- Manual operation
- PMC control function
- Real time custom macro

- Manual handle interrupt

This function cannot be used with the following functions.
- Synchronous / Composite control
- Superimposed control

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
12780								ADE

[Input type] Parameter input
[Data type] Bit path

#0 ADE Deceleration area setting function is :
0: Disabled.
1: Enabled.

12781	Coordinate value of deceleration area (+ direction)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the plus direction in deceleration area boundary.
When the axis moves in the plus direction from this parameter setting value, deceleration is automatically performed so that the speed set to the following parameters is achieved.

	Parameter	
	Rapid traverse	Cutting feed
Deceleration rate	12783	12784

When the multiple axes are moving in deceleration area, the deceleration ratio is calculated for each axis, and the highest deceleration ratio is applied.

12782	Coordinate value of deceleration area (- direction)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the minus direction in deceleration area boundary.
When the axis moves in the minus direction from this parameter setting value, deceleration is automatically performed so that the speed set to the following parameters is achieved.

	Parameter	
	Rapid traverse	Cutting feed
Deceleration rate	12783	12784

When the multiple axes are moving in deceleration area, the deceleration ratio is calculated for each axis, and the highest deceleration ratio is applied.

12783	Deceleration rate of in deceleration area (for rapid traverse)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rapid traverse rate of each axis in the deceleration area.
When this parameter is set to 0, this axis is not decelerated by this function during rapid traverse.

12784	Deceleration rate of in deceleration area (for cutting feed)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the cutting feed rate of each axis in the deceleration area.
When this parameter is set to 0, this axis is not decelerated by this function during cutting feed.

7.1.10 Positioning by Optimum Accelerations

Overview

When rapid traverse is specified during automatic operation, the rapid traverse rate, time constant, and loop gain can be switched according to the travel distance of the block by the function for positioning by optimum accelerations. So, the time required for positioning and in-position check operations can be reduced, resulting in cycle time reduction.
When rapid traverse (G00) is specified during automatic operation, the rapid traverse rate, time constant, and loop gain are switched to one of seven levels according to the travel distance of the block. Parameters are used to specify the rapid traverse rates, time constants, and loop gains corresponding to travel distances.

- Valid command

This function is enabled by the G00 command. In addition, this function is enabled for rapid traverse in a canned cycle, automatic reference position return, and machine coordinate system selection. However, this function is disabled in the cases indicated below, and the feedrate, time constant, and loop gain set in parameters Nos. 1420, 1620, and 1825 respectively are applied.

(1) When automatic reference position return (G28) is specified before reference position establishment
(2) During the Cs contour control mode
(3) During the rigid tapping mode
(4) During the 3-dimensional coordinate system conversion mode

> ⚠ **CAUTION**
> This function is disabled for positioning in a canned cycle.

- Cutting feed

This function is disabled for cutting feed.

- Manual operation
This function is disabled during manual operation.

- Manual intervention
If a manual operation is inserted during automatic operation by halting rapid traverse with feed hold or mode switching, the value of the loop gain is changed to the value set in parameter No. 1825. If the program is restarted, the feedrate, time constant, and loop gain corresponding to the remaining travel distance are used for the rest of the block.

- PMC axis control
This function is disabled for PMC axis control.

- Rapid traverse overlap
When this function is enabled, rapid traverse overlap (bit 4 (RTO) of parameter No. 1601 = 1) is enabled if the same axis is not specified in successive blocks.

```
Example)
    Rapid traverse overlap is performed with the following commands:
    N01 G00 X10.0 ;
    N02 Z-10.0 ;
    Rapid traverse overlap is not performed with the following commands:
    N11 G00 X10.0 Z-10.0 ;
    N12 Z-20.0 ;
```

- Rapid traverse bell-shaped acceleration/deceleration
When rapid traverse bell-shaped acceleration/deceleration is used, T1, T2, and the rapid traverse rate in the Fig. 7.1.10 are switched.

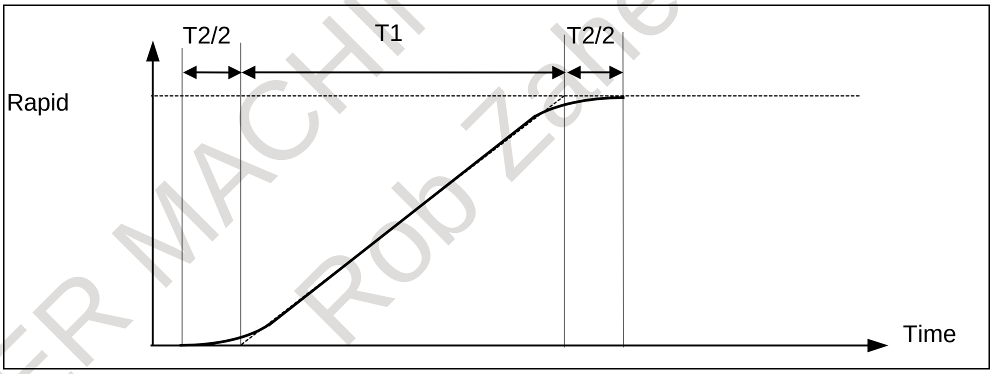


Fig. 7.1.10

- Angular axis control
This function is disabled for an axis to which angular axis control is applied.

- Coordinate system rotation
During the coordinate system rotation mode, switching is performed for the travel distance after coordinate system rotation.

- Scaling (M series)
During the scaling mode, switching is performed for the travel distance after coordinate system rotation. This also applies during the scaling mode.

- Tool radius/Tool nose radius compensation
During the cutter compensation mode or tool-nose radius compensation mode, switching is performed for the travel distance after compensation.

- Linear interpolation type rapid traverse

When linear interpolation type rapid traverse is used (bit 1 (LRP) of parameter No. 1401 = 1), this function is disabled.

- AI contour control

During the AI contour control mode, this function is disabled if look-ahead acceleration/deceleration before interpolation or optimum torque acceleration/deceleration is used for rapid traverse.

- Axis synchronous control

When using this function together with axis synchronous control, enable this function for both of the master axis and slave axis (bit 0 (OAD) of parameter No. 6131 = 1) and set the same feedrate, time constant, and loop gain values for the master axis and slave axis.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6131								OAD

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 OAD The function for positioning by optimum acceleration (seven step switching function of rapid traverse rate by positioning distance and time constant, loop gain at rapid traverse rate in automatic operation) is
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
6132								ILG

[Input type] Parameter input
[Data type] Bit path

#0 ILG In the function for positioning by optimum acceleration, the switch of the loop gain is
0: Enabled. (Parameters Nos. 6181 to 6187 is used.)
1: Disabled. (Parameter No. 1825 is used.)

6136	Distance D1 for level 1 of positioning by optimum acceleration for each axis
6137	Distance D2 for level 2 of positioning by optimum acceleration for each axis
6138	Distance D3 for level 3 of positioning by optimum acceleration for each axis
11230	Distance D4 for level 4 of positioning by optimum acceleration for each axis
11231	Distance D5 for level 5 of positioning by optimum acceleration for each axis
11232	Distance D6 for level 6 of positioning by optimum acceleration for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)

[Valid data range] Refer to the standard parameter setting table (B)

When using the function for switching the rapid traverse rate, time constant, and loop gain according to the positioning distance, this parameter sets the positioning distance for each axis.

> **NOTE**
> 1 To use this parameter, set bit 0 (OADx) of parameter No. 6131 to 1.
> 2 If 0 is set in all of parameters Nos. 6136 to 6138 and 11230 to 11232, this function is disabled.
> 3 The settings must satisfy the following: D1<D2<D3<D4<D5<D6.
> 4 Switching in up to seven steps is possible. When up to four steps are used, for example, set parameters so that expression D1<D2<D3 is satisfied, and set a maximum value (such as +999999.999 mm) for D4, D5, and D6.
> 5 For axes with diameter specification, set a diameter value. If 10.000 mm is set for an axis with diameter specification, for example, switching is made when a movement takes place over a distance of 10.000 mm in diameter.
> 6 In parameters Nos. 6136 to 6138 and 11230 to 11232, set a distance for each axis. Block lengths must not be specified in these parameters.

6161	**Level 1 rapid traverse rate**
6162	**Level 2 rapid traverse rate**
6163	**Level 3 rapid traverse rate**
6164	**Level 4 rapid traverse rate**
6165	**Level 5 rapid traverse rate**
6166	**Level 6 rapid traverse rate**
6167	**Level 7 rapid traverse rate**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Valid data range] Refer to the standard parameter setting table(C)

The rapid traverse rate for each axis is set.

6171	**Level 1 rapid traverse time constant**
6172	**Level 2 rapid traverse time constant**
6173	**Level 3 rapid traverse time constant**
6174	**Level 4 rapid traverse time constant**
6175	**Level 5 rapid traverse time constant**
6176	**Level 6 rapid traverse time constant**
6177	**Level 7 rapid traverse time constant**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

The rapid traverse time constant for each axis is set.

6181	Level 1 servo loop gain
6182	Level 2 servo loop gain
6183	Level 3 servo loop gain
6184	Level 4 servo loop gain
6185	Level 5 servo loop gain
6186	Level 6 servo loop gain
6187	Level 7 servo loop gain

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01/sec
[Valid data range] 1 to 9999

The servo loop gain for each axis is set.
If 0 is set, parameter No. 1825 is used.

6191	Time constant T2 of level 1 bell-shaped acceleration/deceleration in rapid traverse
6192	Time constant T2 of level 2 bell-shaped acceleration/deceleration in rapid traverse
6193	Time constant T2 of level 3 bell-shaped acceleration/deceleration in rapid traverse
6194	Time constant T2 of level 4 bell-shaped acceleration/deceleration in rapid traverse
6195	Time constant T2 of level 5bell-shaped acceleration/deceleration in rapid traverse
6196	Time constant T2 of level 6 bell-shaped acceleration/deceleration in rapid traverse
6197	Time constant T2 of level 7 bell-shaped acceleration/deceleration in rapid traverse

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 512

Time constant T2 of bell-shaped acceleration/deceleration in rapid traverse for each axis is set.

- Table for correspondence between positioning distances and parameter numbers

The correspondence is indicated in the Table 7.1.10.

Table 7.1.10

Level	Positioning distance d	Rapid traverse rate	Rapid traverse time constant T1	Rapid traverse time constant T2	Servo loop gain
1	0 < d ≤ D1	No.6161	No.6171	No.6191	No.6181
2	D1 < d ≤ D2	No.6162	No.6172	No.6192	No.6182
3	D2 < d ≤ D3	No.6163	No.6173	No.6193	No.6183
4	D3 < d ≤ D4	No.6164	No.6174	No.6194	No.6184

Level	Positioning distance d	Rapid traverse rate	Rapid traverse time constant T1	Rapid traverse time constant T2	Servo loop gain
5	D4 < d ≤ D5	No.6165	No.6175	No.6195	No.6185
6	D5 < d ≤ D6	No.6166	No.6176	No.6196	No.6186
7	D6 < d	No.6167	No.6177	No.6197	No.6187

7.1.11 AI Contour Control I and AI Contour Control II

Overview

The AI contour control I and AI contour control II functions are provided for high-speed, high-precision machining. This function enables suppression of acceleration/deceleration delays and servo delays that become larger with increases in the feedrate and reduction of machining profile errors.
There are two types of AI contour control; these two types are referred to temporarily as AI contour control I and AI contour control II. AI contour control I is designed mainly for part machining, and AI contour control II is for machining of successive minute straight lines such as mold machining and so on.
AI contour control represents both AI contour control I and AI contour control II in the rest of this manual.

The functions listed below are valid in the AI contour control mode.

Table 7.1.11 (a) Valid functions

Function	AI contour control I	AI contour control II
Number of blocks read ahead	40 *1 (When G8 is specified: 1)	200 *2 (When G8 is specified: 1)
Look-ahead acceleration/deceleration before interpolation	Linear or bell-shaped acceleration/deceleration	Linear or bell-shaped acceleration/deceleration
Speed control with feedrate difference on each axis	Enabled	Enabled
Speed control with acceleration in circular interpolation	Enabled	Enabled
Speed control with acceleration on each axis	Enabled (When G8 is specified: Not enabled)	Enabled (When G8 is specified: Not enabled)
Smooth speed control	Not enabled	Enabled (When G8 is specified: Not enabled)
Speed control with cutting load	Not enabled	Enabled
Disregard of feedrate command	Not enabled	Enabled

*1 In the system of which maximum controllable paths are more than 2, the number of blocks is 30.
*2 The number of blocks can be expanded to 400 blocks by Look-ahead blocks expansion of AI contour control II.

Format

G05.1 Q1 [R_] ; **AI contour control mode on**
:
G05.1 Q0 ; **AI contour control mode off**
R : Machining condition selecting level (1 to 10)

The AI contour control mode can be controlled also with the formats that have been used for the conventional advanced preview control, high-precision contour control, and AI high-precision contour control functions.

G08 P1 [R_] ;	**AI contour control mode on**
:	
G08 P0 ;	**AI contour control mode off**
R : Machining condition selecting level (1 to 10)	
G05 P10000 [R_] ;	**AI contour control mode on**
:	
G05 P0 ;	**AI contour control mode off**
R : Machining condition selecting level (1 to 10)	

NOTE

1 Always specify G05.1, G08, and G05 in an independent block.
(Do not specify other G codes at the same time.)
2 G05 can be specified only for AI contour control II.
3 The AI contour control mode can also be turned off at a reset.
4 The AI contour control mode can be turned on at the start of automatic operation by setting bit 0 (SHP) of parameter No. 1604.
5 G05.1, G08, and the G05 commands are separate commands respectively. Please turn off the AI contour control mode by the command that turns on the AI contour control mode. AI contour control cannot be turned off by a different command.
6 There are the following conditions when each AI contour control mode on / off is commanded in the multiple.

(1) The AI contour control mode commands are G05, G05.1, and G08 in the order of a high-ranking command (G05 > G05.1 > G08). It is necessary to turn on the AI contour control mode from the subordinate position command sequentially when the multiple commands are executed.

(2) When the AI contour control mode is turned on by the G05P10000 command, even if the AI contour control mode ON/OFF command is executed by G05.1Q1/Q0 command in the mode, the command is ignored.

Example)

G05P10000 AI contour control mode on by G05
G05.1Q1 This command is ignored.
G05.1Q0 This command is ignored.
G05P0 AI contour control mode off by G05

(3) After the AI contour control mode is turned on by the G05P10000 or the G05.1Q1, if the G08 command is executed in the mode, alarm PS5110 “IMPROPER G-CODE (AICC MODE)” occurs.

Example)

G05P10000(G05.1Q1) AI contour control mode on by G05(G05.1)
G08P1(P0) Alarm PS5110 occurs

(4) When the multiple commands are executed, each command that turns on/off the AI contour control mode must be nested.

Example)

G08P1 AI contour control mode on by G08
:
G05.1Q1 AI contour control mode on by G05.1
:
G05P10000 AI contour control mode on by G05
:
G05P0 AI contour control mode off by G05
:
G05.1Q0 AI contour control mode off by G05.1
:
G08P0 AI contour control mode off by G08

⚠ **CAUTION**

Set bit 6 (AP5) of parameter No.11549 to 1 if AI contour control is executed simultaneously by more than 5 paths. If AI contour control is executed simultaneously by more than 5 paths without setting this parameter to 1, sufficient performance might not be obtained.

Explanation

- Look-ahead acceleration/deceleration before interpolation

There are two types of look-ahead acceleration/deceleration before interpolation, the linear acceleration/deceleration type and the bell-shaped acceleration/deceleration type.
Look-ahead bell-shaped acceleration/deceleration before interpolation produces smoother acceleration/deceleration.

- Setting an acceleration

A permissible acceleration for the linear acceleration/deceleration of each axis is set in parameter No. 1660. For bell-shaped acceleration/deceleration, acceleration change time (B) (period of transition from constant speed state (A) to constant acceleration/ deceleration state (C)) is set in parameter No. 1772. In the constant acceleration/deceleration state (C), acceleration/deceleration is performed with the maximum tangential acceleration not exceeding the permissible acceleration of each axis specified in parameter No. 1660.
The acceleration change time specified in parameter No. 1772 is held constant, regardless of the tangential acceleration.

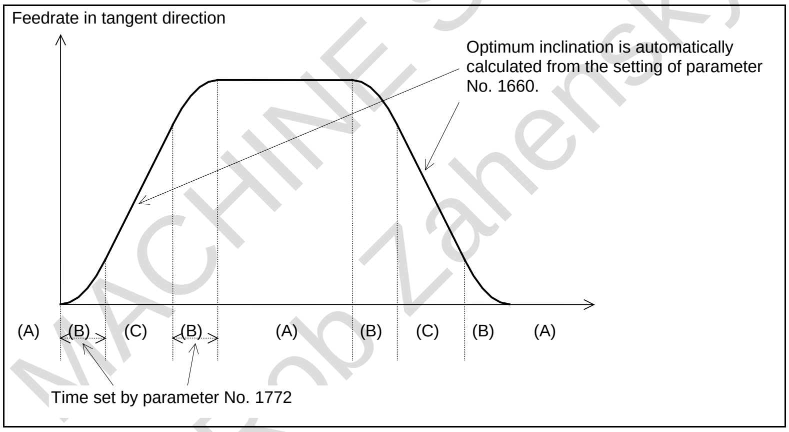


Fig. 7.1.11 (a)

- Method of determining the tangent acceleration

Acceleration/deceleration is performed with the largest tangent acceleration/deceleration that does not exceed the permissible acceleration set for each axis.

(Example)
X-axis permissible acceleration: 1000 mm/sec^2
Y-axis permissible acceleration: 1200 mm/sec^2
Acceleration change time: 20 msec

Program:

```
N1 G01 G91 X20.0 F6000     Move on the X-axis.
G04 X0.01
N2 Y20.0                   Move on the Y-axis.
G04 X0.01
N3 X20.0 Y20.0             Move in the XY direction (at 45 degrees).
```

Since N3 performs interpolation for the X and Y axes in the 45-degree direction, the acceleration of the Y axis is controlled according to the X axis to become 1000 mm/s^2. Therefore, the combined acceleration is 1414 mm/s^2.

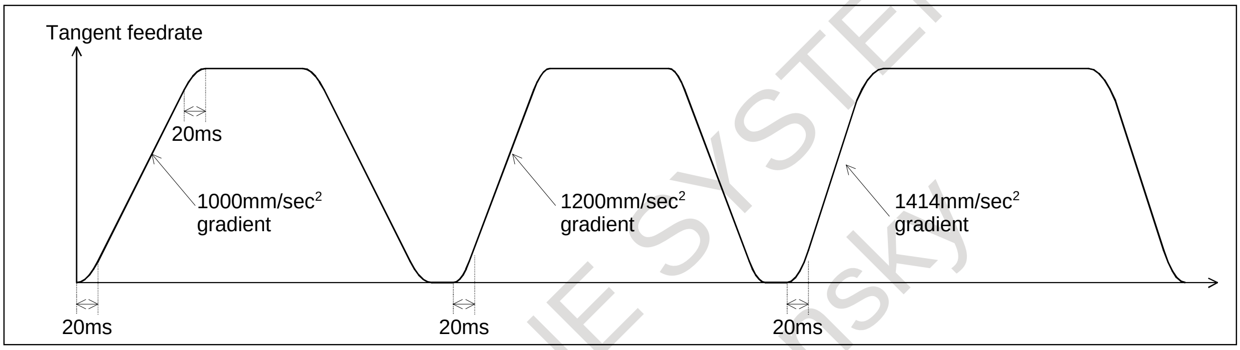


Fig. 7.1.11 (b)

- Acceleration

Acceleration is performed so that the feedrate programmed for a block is attained at the beginning of the block.
When look-ahead acceleration/deceleration before interpolation is valid for multiple blocks, acceleration can be performed across more than one block.

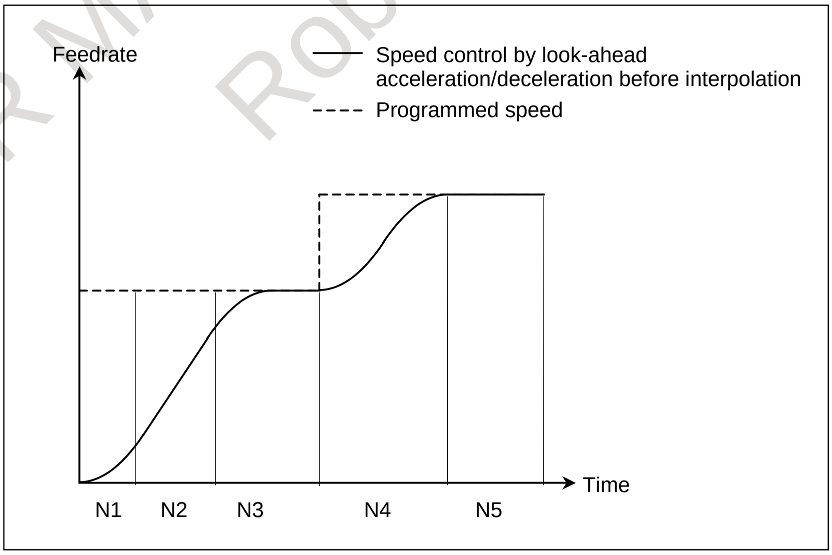


Fig. 7.1.11 (c)

- Deceleration

Deceleration starts in advance so that the feedrate programmed for a block is attained at the beginning of the block.
When look-ahead acceleration/deceleration before interpolation is valid for multiple blocks, deceleration can be performed across more than one block.

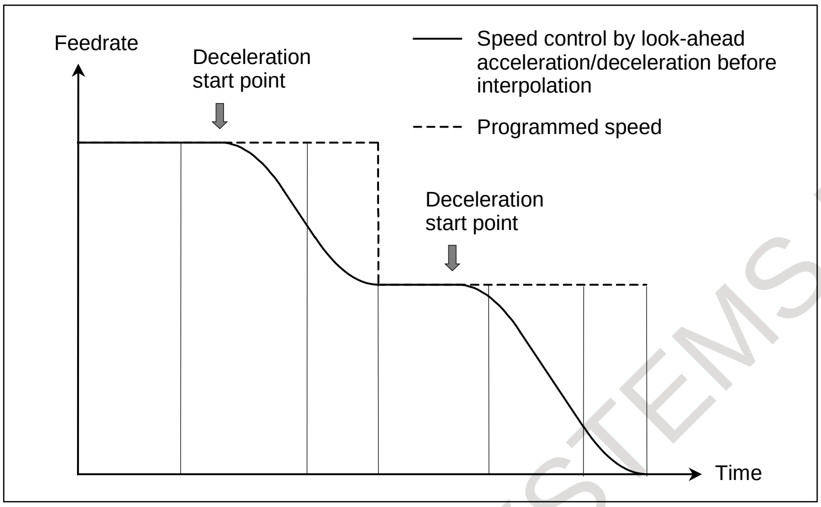


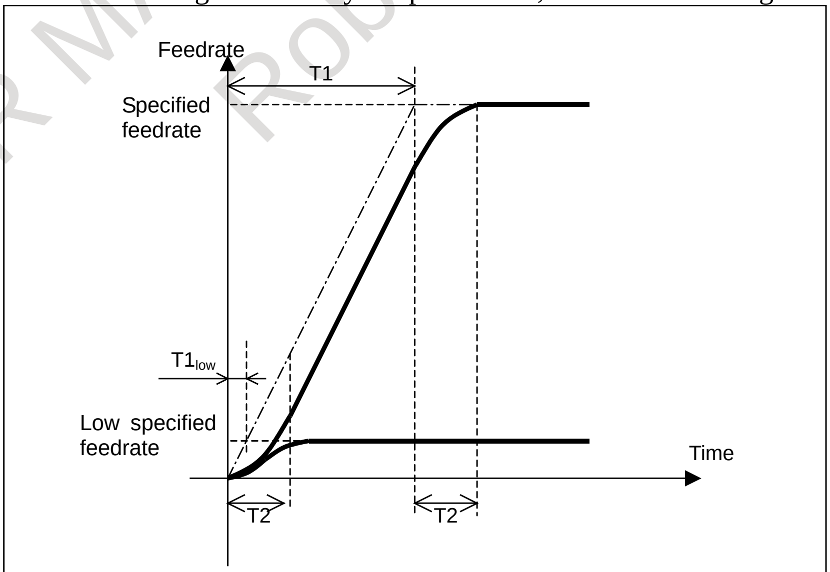
Fig. 7.1.11 (d)

- Deceleration based on a distance

If the total distance of the blocks read ahead becomes shorter than or equal to the deceleration distance obtained from the current feedrate, deceleration starts.
If the total distance of the blocks read ahead during deceleration increases, acceleration is performed.
If the blocks of a small amount of travel are successively specified, deceleration and acceleration may be performed alternately, making the feedrate inconsistent.
To avoid this, decrease the programmed feedrate.

- Function for changing time constant of bell-shaped acceleration/deceleration

Look-ahead bell-shaped acceleration/deceleration before interpolation is performed according to the acceleration and acceleration change time set by the parameters, as shown in the figure below.



T1 : Time obtained from specified feedrate and specified acceleration (specified feedrate/acceleration (parameter No. 1660))
T2 : Acceleration change time (parameter No. 1772)

Fig. 7.1.11 (e)

Here, the acceleration change time (T2) remains constant regardless of the specified feedrate, while the acceleration time for the linear section (T1), which is determined by acceleration, varies with the specified feedrate. If T1 becomes shorter than T2 when the specified feedrate is low, linear acceleration/deceleration not achieving the specified acceleration results, as shown in the figure below.

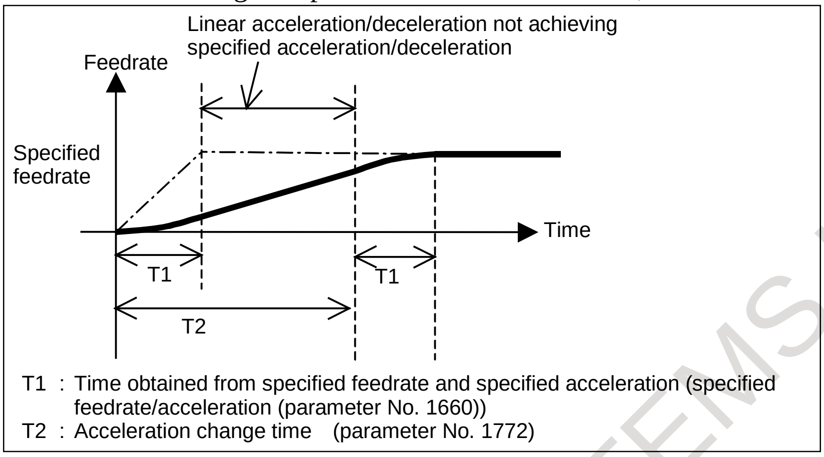


Fig. 7.1.12 (f)

In such a case, set bit 3 (BCG) of parameter No. 7055 to 1. Then, the internal acceleration and vector time constant of look-ahead acceleration/ deceleration before interpolation are changed to make the acceleration/deceleration pattern as close as possible to the optimum look-ahead bell-shaped acceleration/deceleration before interpolation based on a specified acceleration/deceleration reference speed, and so acceleration/deceleration time is reduced.

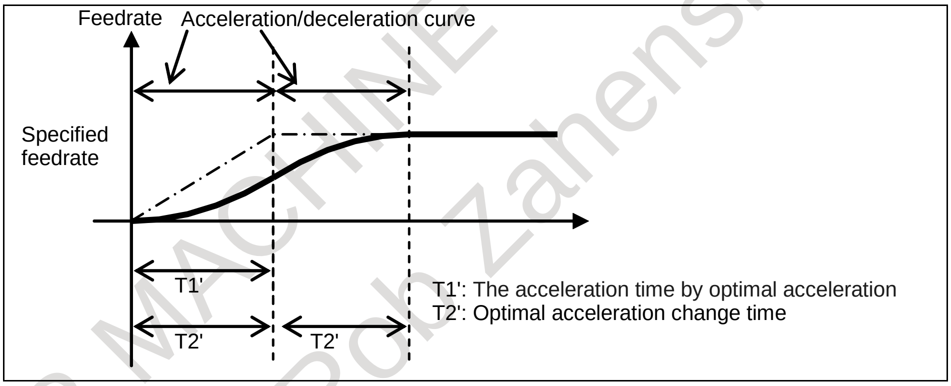


Fig. 7.1.11 (g)

There are three methods for specifying the acceleration/deceleration reference speed.

(1) Specifying the speed using an F in a G05.1 Q1 block
(2) Setting the speed on parameter No. 7066
(3) Setting the speed specified with the F command issued at the start of cutting as the reference speed

When F is specified in a G05.1Q1 block, the specified feedrate is assumed to be the acceleration/deceleration reference speed. This command can be used only in the feed per minute mode.
If no F command is specified in a G05.1Q1 block, the feedrate specified in parameter No. 7066 is assumed to be the acceleration/deceleration reference speed. If 0 is set in parameter No. 7066, the F command specified in the cutting start block is assumed to be the acceleration/deceleration reference speed.

> **NOTE**
> When function for changing time constant of look-ahead bell-shaped acceleration/deceleration before interpolation is enabled (bit 3 (BCG) of parameter No. 7055 is set to 1), be sure to specify acceleration rate of the reference axis.
> If acceleration rate of the reference axis is not specified, optimum acceleration rate is not obtained.

- Acceleration/deceleration after cutting feed interpolation in AI contour control mode

For cutting feed in AI contour control mode, in addition to look-ahead acceleration/deceleration before interpolation, acceleration/deceleration after interpolation can be enabled.
The time constant of acceleration/deceleration after interpolation is set to the parameter (No.1769).
Type of acceleration/deceleration after interpolation is selected by bit 6 (LS2) and bit 3 (BS2) of parameters No. 1602.
Using exponential acceleration/deceleration: LS2 is set to 0 and BS2 is set to 0.
Using linear acceleration/deceleration: LS2 is set to 1 and BS2 is set to 0.
Using bell-shaped acceleration/deceleration: BS2 is set to 1.
If shape of acceleration/deceleration after interpolation is exponential acceleration/deceleration, FL feedrate for each axis is set to parameter No. 1763.

Feedrate control function

In AI contour control mode, the feedrate is automatically controlled by the look-ahead of blocks.
The feedrate is determined using the following conditions. If the specified feedrate exceeds the determined feedrate, look-ahead acceleration/ deceleration before interpolation is performed to achieve the determined feedrate.
(1) Feedrate changes on each axis at a corner and the permissible feedrate change that has been set
(2) Expected acceleration on each axis and the permissible acceleration that has been set
(3) Cutting load that is expected from the travel direction on the Z-axis

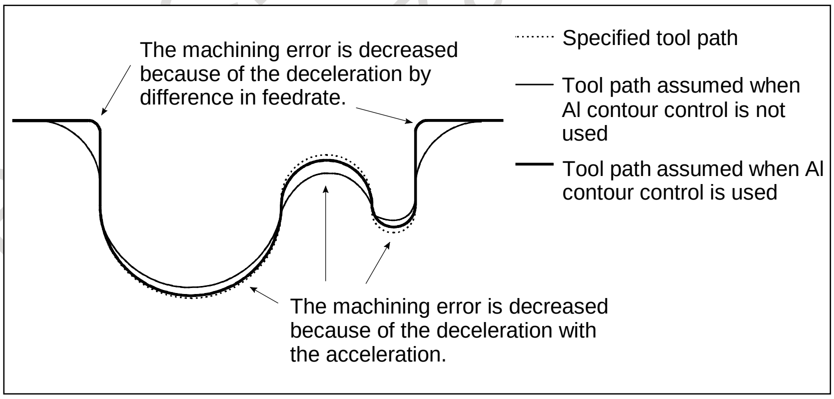


Fig. 7.1.11 (h)

For details, see the explanation of each function.

- Speed control based on the feedrate difference on each axis at a corner

By using the speed control based on the feedrate difference on each axis at a corner, if a feedrate change occurs on an axis on each axis at a corner, the feedrate is determined so that any feedrate difference exceeding the permissible feedrate difference on that axis that has been set for parameter No. 1783 does not occur, and deceleration is automatically performed.

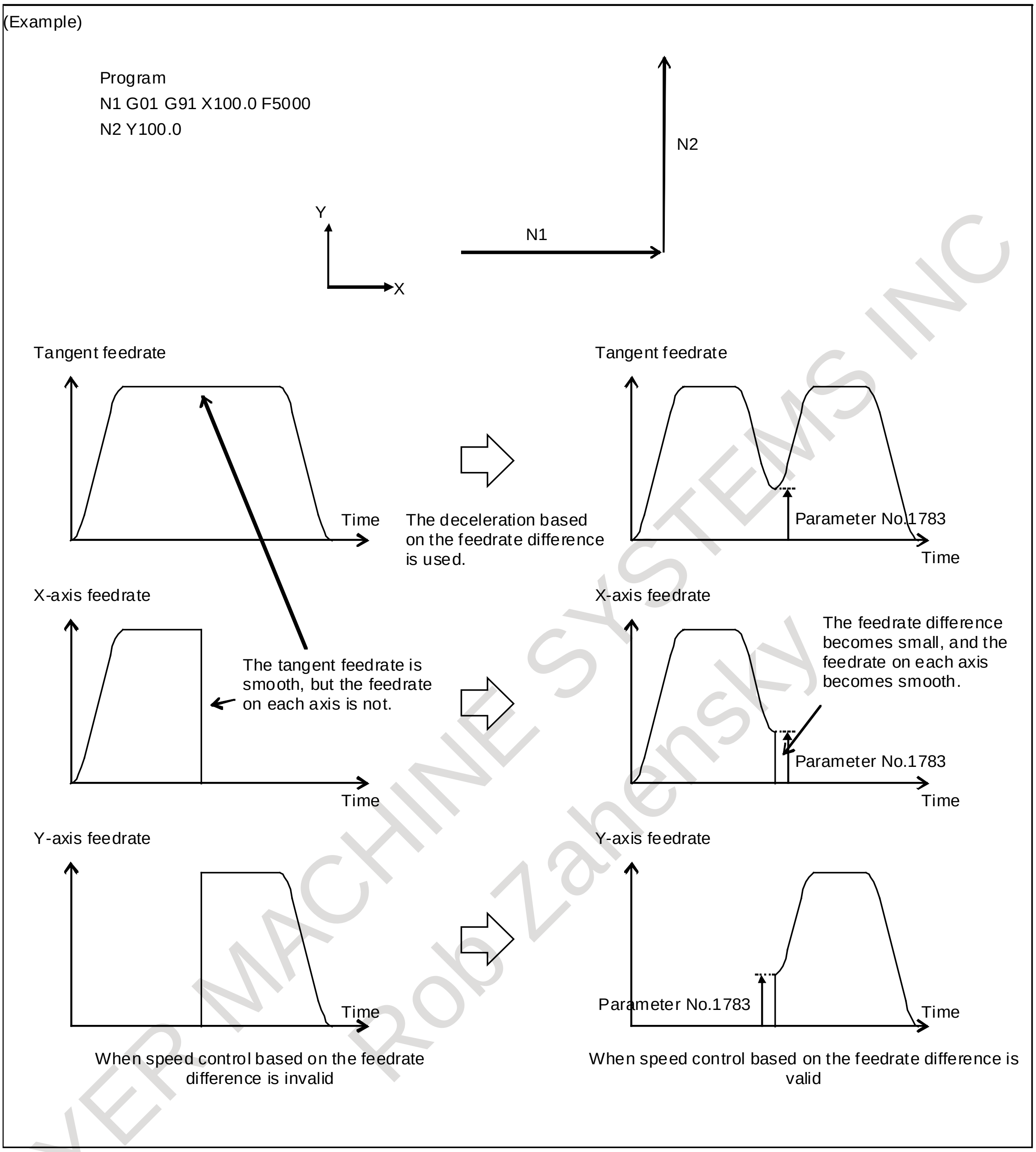


Fig. 7.1.11 (i)

The method of deceleration based on the feedrate difference differs depending on the setting made for parameter FNW (bit 6 of No. 19500).
If parameter FNW is set to 0, the largest feedrate that does not exceed the permissible feedrate difference set for parameter No. 1783 is assumed to be the deceleration feedrate.
In this case, the deceleration feedrate differs if the travel direction differs, even if the shape is the same.

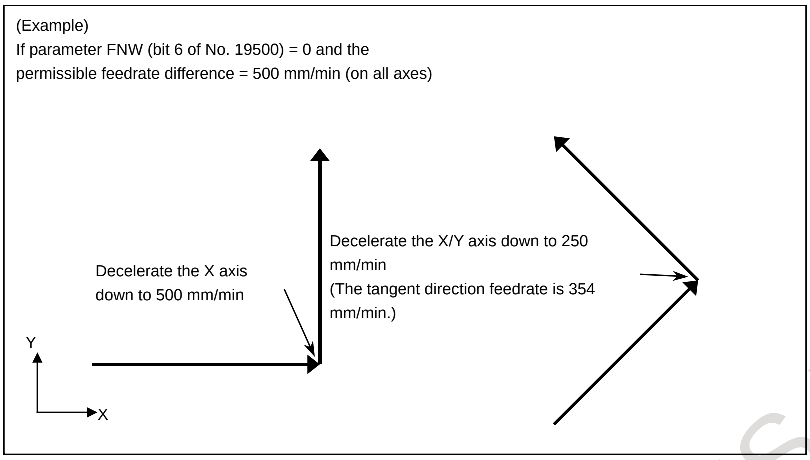


Fig. 7.1.11 (j)

In the left-side example in the figure above, the X axis is inverted at the corner from the position direction to the negative direction, and deceleration is performed so that the feedrate difference becomes 500 mm/min. In other words, the feedrate is 250 mm/min both when the axis moves in the position direction and when it moves in the negative direction. As a result, the tangent direction feedrate becomes 354 mm/min.

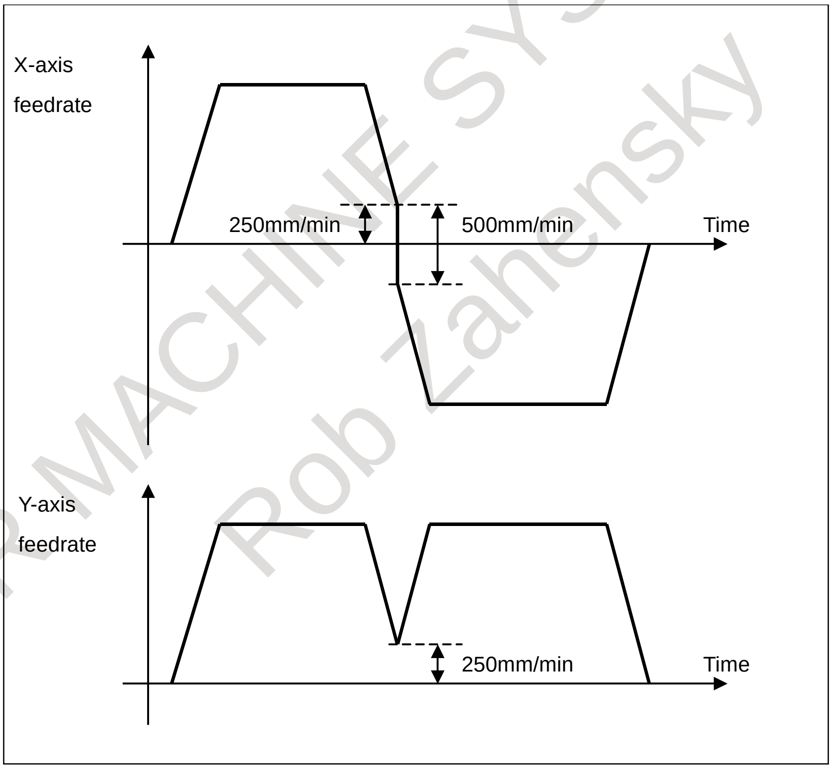


Fig. 7.1.11 (k)

If parameter FNW is set to 1, the feedrate is determined not only with the condition that
the permissible feedrate difference and permissible acceleration on each axis are not exceeded, but also that the deceleration feedrate is constant regardless of the travel direction if the shape is the same.
If this parameter is set to 1, the deceleration feedrate determined with the feedrate difference may be up to 30% lower than that determined if this parameter is set to 0.

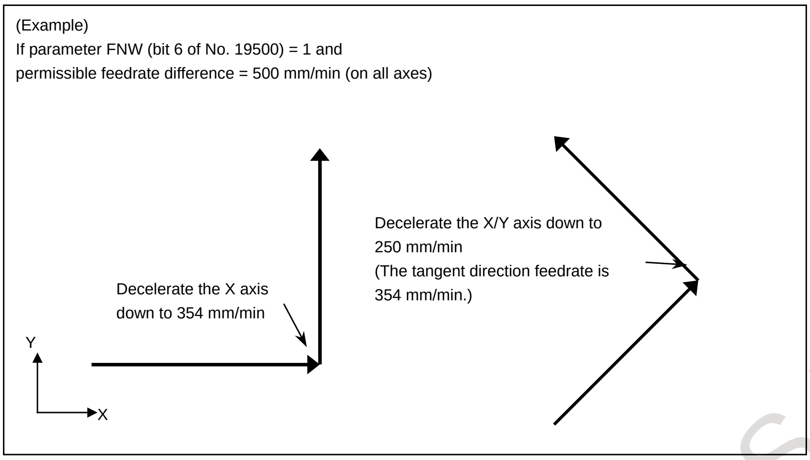


Fig. 7.1.11 (l)

- Speed control with acceleration in circular interpolation

When high-speed cutting is performed in circular interpolation, helical interpolation, or spiral interpolation, the actual tool path has an error with respect to the programmed path. In circular interpolation, this error can be approximated from the equation given below.

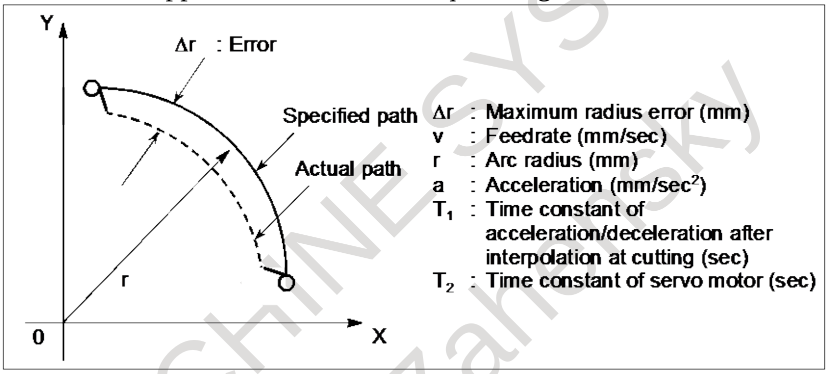


Fig. 7.1.11 (m)

$$\Delta r = \frac{1}{2}(T_1^2 + T_2^2)\frac{v^2}{r} = \frac{1}{2}(T_1^2 + T_2^2)\cdot a \quad \text{.......... (Equation 1)}$$

In actual machining, permissible error Δr is given, so the maximum permissible acceleration a (mm/sec^2) in equation 1 is determined.

Expression 1 is an expression when type of acceleration/deceleration after interpolation is exponential acceleration/deceleration. The radius error in the case of linear acceleration/deceleration after interpolation is smaller than in case of exponential acceleration/deceleration by a factor of 12, excluding any error caused by a servo motor time constant. The radius error in the case of bell-shaped acceleration/deceleration after interpolation is a half of linear acceleration/deceleration after interpolation.

When a specified feedrate causes the radial error from an arc having a programmed radius to exceed the permissible error, speed control with acceleration in circular interpolation automatically clamps the arc-cutting feedrate by using parameter settings.

Let the permissible acceleration calculated from the permissible acceleration set for each axis be A. Then, maximum permissible feedrate v with programmed radius r is expressed as follows:

$$v = \sqrt{A \cdot r} \quad \text{.......... (Equation 2)}$$

If a specified feedrate exceeds feedrate v obtained from equation 2, the feedrate is clamped at feedrate v automatically.

The permissible acceleration is specified in parameter No. 1735. If there is a difference in permissible acceleration between two axes for circular interpolation, the lower acceleration is regarded as the permissible acceleration.

If the radius of an arc is small, too small value can be calculated as deceleration v. In such a case, the lower feedrate limit can be set in parameter No. 1732 to prevent the feedrate from being decreased too much.

- Speed control with the acceleration on each axis

When consecutive small lines are used to form a curve, as in the example shown in the Fig. 7.1.11 (n), the feedrate differences on each axis at the individual corners are not very large. Thus, deceleration with the feedrate differences is not effective. Consecutive small feedrate differences, however, cause a large acceleration on each axis, as a whole.
In such a case, deceleration can be performed to reduce the impact on the machine and the machining error caused by too large an acceleration. The deceleration feedrate is determined to be the feedrate that does not cause the acceleration on each axis to exceed the permissible acceleration set for parameter No. 1737.
The deceleration feedrate is determined for each corner. The actual feedrate is the smaller of the deceleration feedrate determined at the start point of the block and that determined at the end point.
Depending on the specified figure, a very low deceleration feedrate may be calculated. In such a case, the lower feedrate limit can be set in parameter No. 1738 to prevent the feedrate from being decreased too much.

In the following example, the acceleration (gradient of the broken line in the feedrate graph) at too large at corners N2 to N4 and N6 to N8 and, therefore, deceleration is performed.

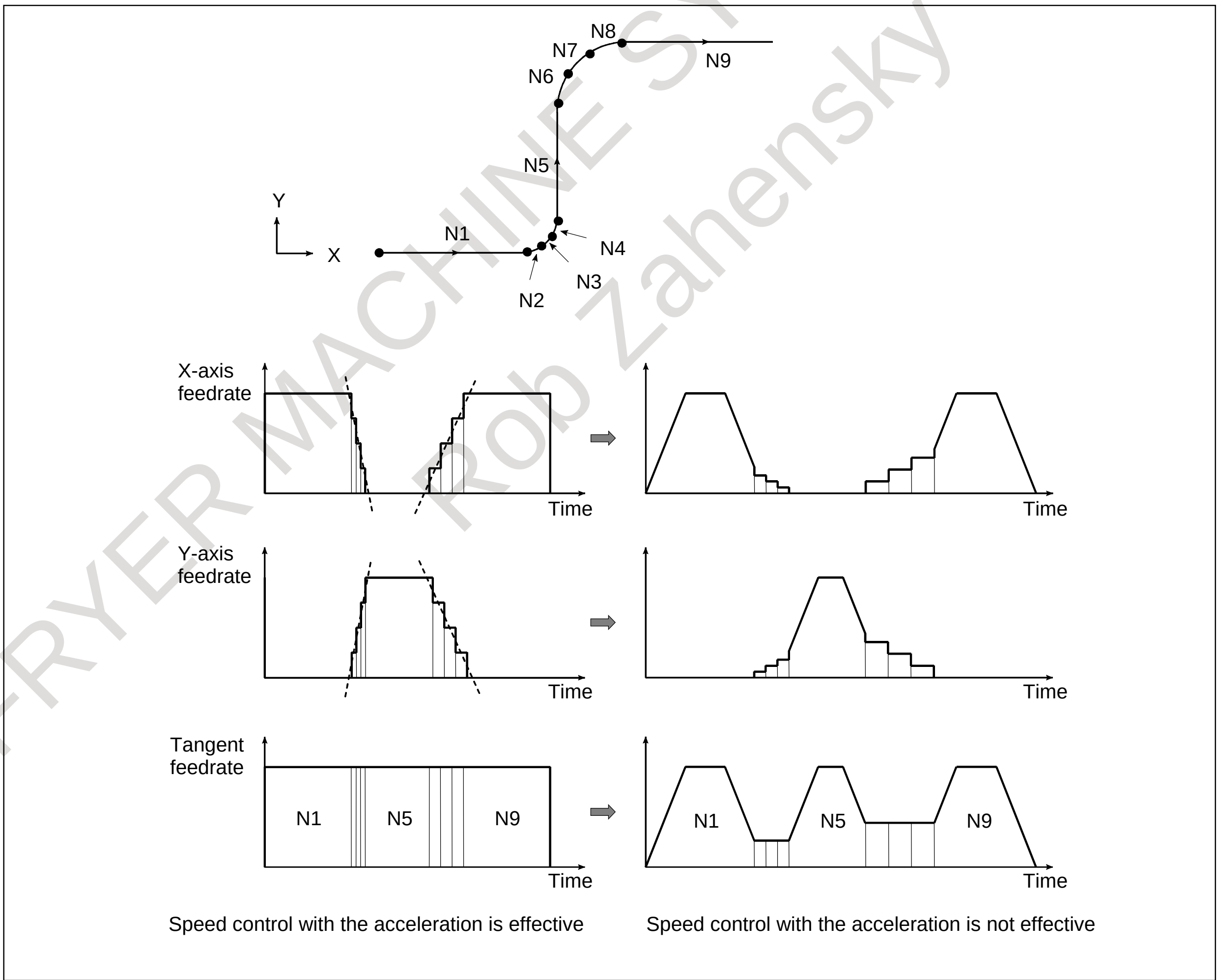


Fig. 7.1.11 (n)

The method of determining the feedrate with the acceleration differs depending on the setting of parameter FNW (bit 6 of No. 19500).

If parameter FNW is set to 0, the highest feedrate that does not cause the permissible acceleration set for parameter No. 1737 to be exceeded is assumed to be the deceleration feedrate. In this case, the deceleration feedrate differs depending on the travel direction even if the shape is the same, as shown in the figure below.

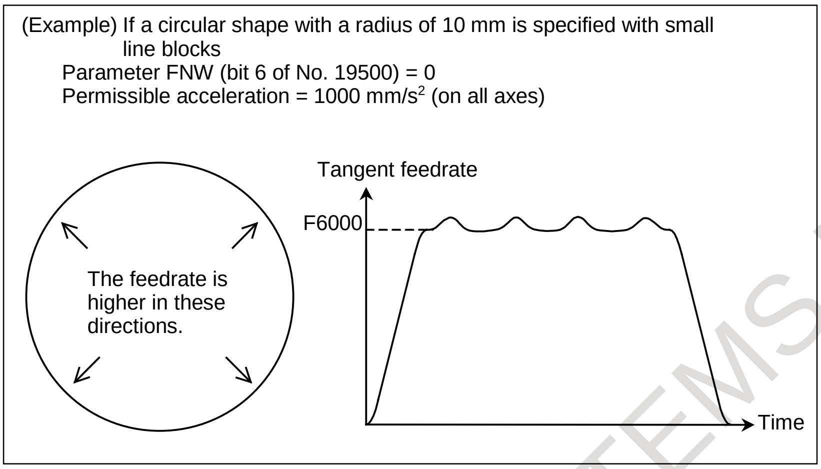


Fig. 7.1.11 (o)

If parameter FNW is set to 1, the feedrate is determined with not only the condition that the permissible acceleration on each axis is not exceeded but also the condition that the deceleration feedrate is constant regardless of the travel direction if the shape is the same.
If this parameter is set to 1, the deceleration feedrate determined with the feedrate difference or acceleration may be up to 30% lower than that determined if this parameter is set to 0.

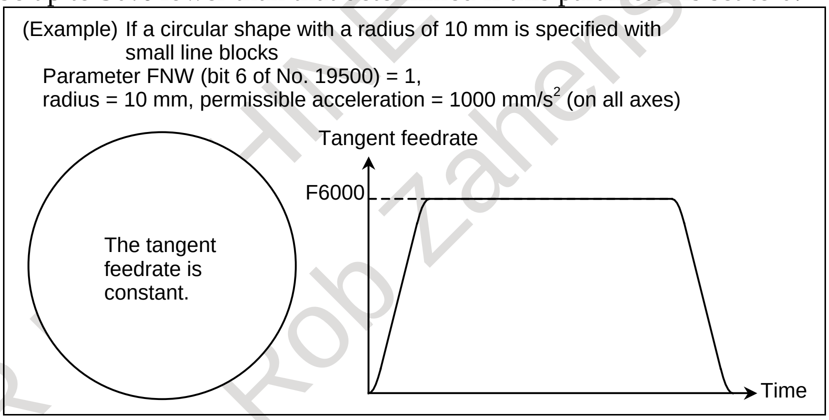


Fig. 7.1.11 (p)

> **NOTE**
> In circular interpolation, the tangent feedrate is constant regardless of the setting of the parameter.

When this function is effective, the feedrate might be clamped as well as the travel distance in the block is small even if the travel distance in the block is long. Speed control with the acceleration is invalidated in the block longer than the block length set to parameter No. 19518 if bit 1 (HNG) of parameter No. 19517 is set to 1.

- Smooth speed control

In speed control with acceleration, the smooth speed control function recognizes the entire figure from preceding and following blocks including blocks read ahead to make a smooth feedrate determination.
When a curve is specified with successive minute straight lines, programmed values are rounded to the least input increment before issued, so the machining profile is approximated with a broken line.

When the feedrate is determined with acceleration in an ordinary manner, an optimum feedrate is automatically calculated exactly for a programmed figure, so a large acceleration may result depending on the command, which can lead to deceleration.
In such a case, the use of smooth speed control enables speed control by recognizing the entire figure, which provides smooth speed control while suppressing local deceleration, therefore increasing the feedrate.

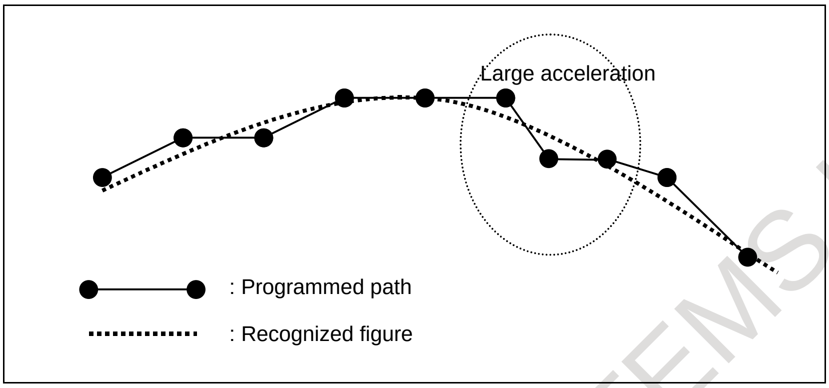


Fig. 7.1.11 (q)

Also for a part of a programmed figure in which a large acceleration would be required, the acceleration is obtained based on the figure recognized from multiple blocks, and the feedrate is determined so that the acceleration is within the permissible acceleration set in parameter No. 1737.

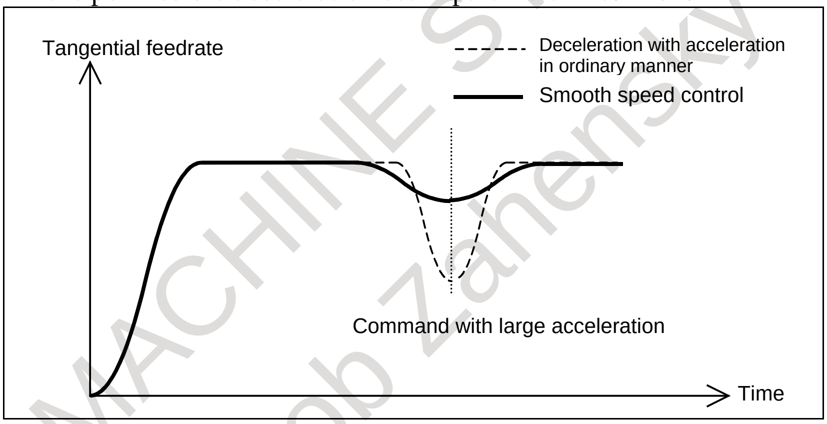


Fig. 7.1.11 (r)

Smooth speed control obtains the acceleration by using the figure recognized from the preceding and following blocks including blocks read ahead, so smooth speed control is enabled even in parts in which the acceleration increases.
Smooth speed control is enabled under the following conditions:
(1) Speed control with acceleration is enabled in the AI contour control II mode.
(2) Successive linear interpolation commands are specified.
(3) Bit 0 (HPF) of parameter No. 19503 is set to 1.

> ⚠ **CAUTION**
> When smooth speed control is used, the feedrate in a certain figure such as a corner may become larger than the feedrate obtained by ordinary speed control with acceleration. For corners, set parameter No. 1783, which is the permissible feedrate difference parameter for speed control with the feedrate difference at corners, to perform appropriate deceleration by speed control with the corner feedrate difference.

When this function is effective, the feedrate might be clamped as well as the travel distance in the block is small even if the travel distance in the block is long. Smooth speed control is invalidated in the block longer than the travel distance set to parameter No. 19518 if bit 1 (HNG) of parameter No. 19517 is set to 1.

- Deceleration with the cutting load

Usually, the cutting resistance produced when machining is performed with the bottom of the cutter as the tool lowers along the Z-axis is greater than the cutting resistance produced when machining is performed with the side of the cutter as the tool rises along the Z-axis. Therefore, deceleration is required.
In AI contour control II, the tool travel direction on the Z-axis is used as a condition for calculating the machining feedrate.
This function is enabled when parameter ZAG (bit 4 of No. 8451) is set to 1.

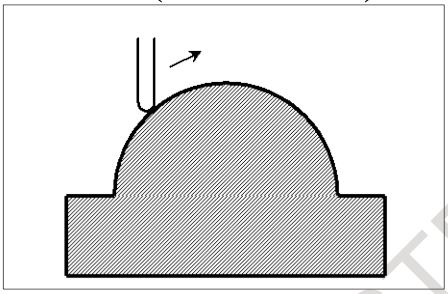


Fig. 7.1.11 (s) During ascent on the Z-axis

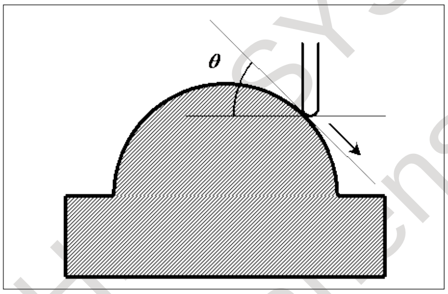


Fig. 7.1.11 (t) During descent on the Z-axis

The descent angle θ during descent on the Z-axis (angle formed by the XY plane and the tool center path) is as shown in the Fig. 7.1.11 (s). The descent angle is classified into four areas, and the override values for the individual areas are set for the following parameters:

Parameter No. 8456 for area 2
Parameter No. 8457 for area 3
Parameter No. 8458 for area 4

For area 1, however, no parameter is available, and an override of 100% is used at all times. The feedrate obtained according to other feedrate control is multiplied by the override value of the area to which descent angle θ belongs.

Area1 $0° \le \theta < 30°$
Area2 $30° \le \theta < 45°$
Area3 $45° \le \theta < 60°$
Area4 $60° \le \theta < 90°$

The feedrate can be overridden with an inclination by setting bit 1 (ZG2) of parameter No. 19515 to 1. In this case, specify the override value for area 1 in parameter No. 19516.

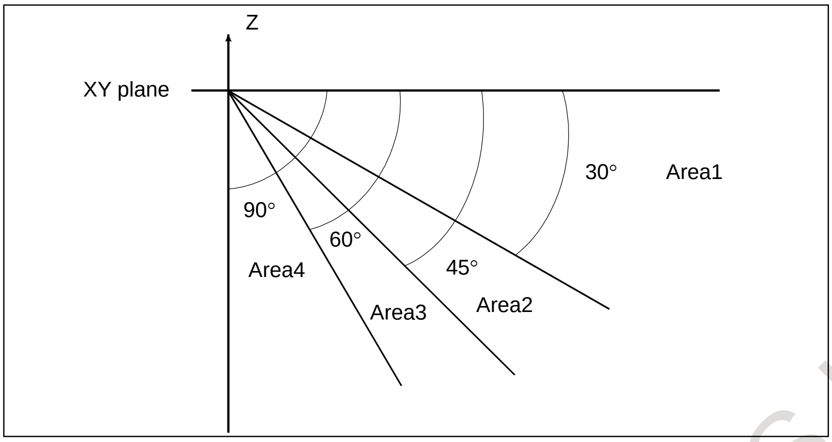


Fig.7.1.11 (u)

> ⚠ **CAUTION**
> 1 The speed control with the cutting feed is effective only when the tool is parallel with the Z-axis. Thus, it may not be possible to apply this function, depending on the structure of the machine used.
> 2 In the speed control with the cutting feed, the travel direction on the Z-axis is determined with the appropriate NC command. If, therefore, manual intervention is performed on the Z-axis with manual absolute on, or if a mirror image is applied on the Z-axis, the direction on the Z-axis cannot be determined. When using the speed control with the cutting load, do not use these functions.
> 3 When performing 3-dimensional coordinate system conversion, determine the descent angle on the Z-axis using the converted coordinate system.
> 4 Speed control with the cutting load is enabled for all interpolations in the AI contour control II mode. This function, however, can be made valid only for linear interpolations by setting bit 4 (ZOL) of parameter No. 19503 to 1.

- Ignoring feedrate commands

In a block in which AI contour control II is enabled, all feedrate commands (F commands) can be ignored by setting parameter NOF (bit 7 of No. 8451).
The term feedrate commands, as used here, refer to the following commands:

(1) Modal F commands before the block in which AI contour control II is enabled
(2) F commands and modal F commands in the block in which AI contour control II is enabled

When the feedrate commands are ignored, it is assumed that the upper feedrate limit specified for parameter No. 8465 is specified.
When a rotation axis is singly specified, it is assumed that the upper feedrate limit specified for parameter No. 8466 is specified.
Note, however, that any issued F commands and modal F commands are stored within the CNC.
Thus, in a block in which AI contour control II changes from the enabled state to the disabled state, the modal values of the F commands described in (1) and (2) described above are used as modal F commands, instead of the modal values of the F commands calculated by AI contour control II.

- Another example of determining the feedrate

If a specified feedrate exceeds the upper feedrate limit of AI contour control (in parameter No. 8465, or parameter No. 8466 (when a rotation axis is singly specified))), the feedrate is clamped at the upper feedrate. The upper feedrate limit is clamped at the maximum cutting feedrate (parameter No. 1432) for acceleration/deceleration before interpolation

Status display

Depending on the control mode, one of the status indications shown below blinks at the lower right of the screen.

Table7.1.11 (b)

	AI contour control I	AI contour control II
Status display	AICC 1	AICC 2

- In the case of AI contour control I, the character string to be displayed can be changed using parameter Nos. 3241 to 3247.
- In the case of AI contour control II, the character string to be displayed can be changed using parameter Nos. 3251 to 3257..

The advanced preview feed forward function when the AI contour control mode is off

The advanced preview feed forward function is enabled when bit 1 (FEED) of parameter No. 2005 for enabling the advanced preview feed forward function is set to 1 and a value is set in parameter No. 2092, regardless of whether AI contour control is turned on or off.
By setting bit 2 (AOFF) of parameter No. 1611 to 1, the advanced preview feed forward function is disable when command for turning on AI contour control is not specified yet.

Example of the advanced preview feed forward function turned on/off

(1) Bit 0 (SHP) of parameter No. 1604 is set to 0
When bit 0 (SHP) of parameter No. 1604 is set to 0, the AI contour control mode is not turned on if the command for AI contour control ON is not specified. Program operation in this state is explained below.

```
O0001;
N01 G90 G01 X50.0 Y50.0 F1000. ;
N02 G05.1 Q1 ;
N03 X100.0 Y100.0 ;
N04 G05.1 Q0 ;
N05 X150.0 Y150.0 ;
N06 M02 ;
```

(a) When bit 2 (AOFF) of parameter No. 1611 is set to 0
The advanced preview feed forward function is enabled in the N01, N03, and N05 blocks.
(b) When bit 2 (AOFF) of parameter No. 1611 is set to 1
The advanced preview feed forward function is enabled in the N03 block.
The advanced preview feed forward function is disabled in the N01 and N05 blocks.

(2) Bit 0 (SHP) of parameter No. 1604 is set to 1
When bit 0 (SHP) of parameter No. 1604 is set to 1, the program operates in the same way as when G05.1Q1 is specified at automatic operation start time (from the reset state). Program operation from the reset state is explained below.

```
O0001;
N01 G90 G01 X50.0 Y50.0 F1000. ;
N02 G05.1 Q1 ;
N03 X100.0 Y100.0 ;
N04 G05.1 Q0 ;
N05 X150.0 Y150.0 ;
N06 M99;
```

(a) When bit 2 (AOFF) of parameter No. 1611 is set to 0
The advanced preview feed forward function is enabled in all movement blocks of N01, N03, and N05.

(b) When bit 2 (AOFF) of parameter No. 1611 is set to 1
Immediately after automatic operation is started, the advanced preview feed forward function is enabled in the N01 and N03 blocks and the advanced preview feed forward function is disabled in the N05 block.
If N06 specifies M99 to return to the start of the program, the advanced preview feed forward function is disabled in the N01 block because N04 turns off the AI contour control mode.

(3) When none of the AI contour control is specified
G05.1Q1 cannot be specified. This state is equivalent to the state where G05.1Q0 is specified, that is, AI contour control is off at all times. Program operation in this state is described below.

```
O0002;
N01 G90 G01 X50.0 Y50.0 F1000. ;
N02 X100.0 Y100.0 ;
N03 X150.0 Y150.0 ;
N04 M02 ;
```

(a) When bit 2 (AOFF) of parameter No. 1611 is set to 0
The advanced preview feed forward function is enabled in the N01, N02, and N03 blocks.
(b) When bit 2 (AOFF) of parameter No. 1611 is set to 1
The advanced preview feed forward function is disabled in the N01, N02, and N03 blocks.

Limitation

- Conditions for temporarily canceling the AI contour control mode

If one of the commands listed below is issued in the AI contour control mode, the AI contour control mode is canceled temporarily.
If the system becomes ready for AI contour control after it is canceled, the AI contour control mode is restored automatically.

- Positioning (rapid traverse)
- Single-direction positioning
- Spindle positioning
- Rigid tapping
- Threading cycle (threading and multiple repetitive threading)
- Electronic gear box
- Superimposed control
- When no move command is specified
- One-shot G code other than the following:
  Tool offset
  Cutter compensation vector retention
  Cutter compensation corner rounding
  Exact stop

Example

```
O0010
…
G5.1 Q1;
G01
…
X1.0Y2.0Z3.0;   } AI contour control mode
M220;           Start superimposed control     } AI contour control mode is
…                                              } temporarily canceled.
M221;           Cancel superimposed control
X2.0Y2.0Z4.0;   } Automatically return the system to AI contour control mode
…               } AI contour control mode
X4.0Y1.0Z2.0;
G5.1 Q0;
…
M30;
```

(Note The way to specify synchronous, composite, and superimposed controls differ from one machine tool builder to another. For details, refer to the manual issued by the machine tool builder.)

- Functions that cannot be specified in the AI contour control mode

In the AI contour control mode, the functions listed below cannot be specified. Alarm PS5110 "IMPROPER G-CODE (AICC MODE)" occurs. Before specifying these functions, turn off the AI contour control mode; after the command ends, turn on the mode again.

- Threading
- Circular threading
- Variable lead threading

To specify threading, circular threading, and variable lead threading in the AI contour control mode, set THA (bit 1 of parameter No. 1611). However, the AI contour control mode is automatically cancelled.

- Modal G codes usable when AI contour control is specified

In a modal G code state listed below, AI contour control can be specified.
Do not specify AI contour control in modal states other than these.

G13.1	: Polar coordinate interpolation mode cancel
G15	: Polar coordinate command cancel
G40	: Tool radius compensation cancel
G40.1	: Normal direction control cancel
G50	: Scaling cancel
G50.1	: Programmable mirror image cancel
G50.2	: Polygon turning cancel
G64	: Cutting mode
G67	: Macro modal call cancel
G69	: Coordinate system rotation/3-dimensional coordinate system conversion cancel
G80	: Canned cycle cancel
G97	: Constant surface speed control cancel

- Retrace

When the reverse execution is started in AI contour control mode, a reverse execution ends immediately depending on a program then the backward movement is not possible.

During reverse execution and forward reexecution, the feedrate clamp function by acceleration under AI contour control is disabled.

- Manual handle interrupt

Manual handle interrupt is available in AI contour control mode.

- DNC operation

AI contour control is available during DNC operation.

Signal

AI contour control mode signal AICC <Fn062.0>

[Classification] Output signal

[Function] This signal indicates that the system is in AI contour control mode.

[Output cond.] This signal is set to 1 when the cutting command or other conditions for AI contour control is met in AI contour control mode.
This signal is set to "0" in the following cases.

(1) In the automatic operation suspend state
(2) In the automatic operation stop state
(3) When rapid traverse or other conditions under which AI contour control is automatically cancelled are met
(4) When AI contour mode is cancelled

> **NOTE**
> Even if the conditions for automatically canceling AI contour control, such as rapid traverse, are satisfied, the signal is output if bit 1 (AIR) of parameter No. 1612 is 1 (status display in AI contour control mode and the AI contour control mode signal are enabled).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F062								AICC

Parameter

1432	Maximum cutting feedrate for all axes in the acceleration/deceleration before interpolation

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set a maximum cutting feedrate for each axis in the look-ahead acceleration/deceleration before interpolation mode such as AI contour control. When this parameter is set to 0 or the look-ahead acceleration/deceleration before interpolation mode is not set, the maximum cutting feedrate set in parameter No. 1430 is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1602		LS2			BS2			

[Input type] Parameter input
[Data type] Bit path

#3 BS2 Acceleration/deceleration after interpolation for cutting feed in a mode of look-ahead acceleration/deceleration before interpolation such as the AI contour control mode:

0: Exponential acceleration/deceleration or linear acceleration/ deceleration is used. (The setting of bit 6 (LS2) of parameter No. 1602 is followed.)

1: Bell-shaped acceleration/deceleration is used.

#6 LS2 Acceleration/deceleration after interpolation for cutting feed in a mode of look-ahead acceleration/deceleration before interpolation such as the AI contour control mode:
0: Exponential acceleration/deceleration is used.
1: Linear acceleration/deceleration is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1604								**SHP**

[Input type] Parameter input
[Data type] Bit path

#0 SHP When automatic operation is started, the state equivalent to the specification of G5.1Q1 for AI contour control is:
0: Not set
1: Set

> **NOTE**
> It is necessary to reset CNC once after this parameter is changed.

	#7	#6	#5	#4	#3	#2	#1	#0
1611						**AOF**		

[Input type] Parameter input
[Data type] Bit path

#2 AOF When AI contour control mode is off and the parameter of the advanced-preview feed forward function is valid, the advanced-preview feed forward function is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
1612							**AIR**	

[Input type] Parameter input
[Data type] Bit path

#1 AIR The status display and mode signal in AI contour control mode is:
0: Enabled only when the conditions for executing AI contour control are satisfied.
1: Enabled always in AI contour control mode.
If this parameter is set to 1, in-mode blinking display and the AI contour control mode signal AICC <Fn062.0> remain enabled in the AI contour control mode.

> **NOTE**
> The parameter is invalid during a stop on feed hold or a single-block stop.

1660	**Maximum allowable acceleration rate in acceleration/deceleration before interpolation for each axis**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Set a maximum allowable acceleration rate in look-ahead acceleration/ deceleration before interpolation for each axis.
If a value greater than 100000.0 is set, the value is clamped to 100000.0.
If 0 is set, the specification of 100000.0 is assumed. If 0 is set for all axes, however, look-ahead acceleration/deceleration before interpolation is not performed.
If a maximum allowable acceleration rate set for one axis is greater than a maximum allowable acceleration rate set for another axis by a factor of 2 or more, the feedrate at a corner where the direction of travel abruptly changes can decrease temporarily.

1732	Minimum allowable feedrate for the deceleration function based on acceleration in circular interpolation

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
With the deceleration function based on acceleration in circular interpolation, an optimum feedrate is automatically calculated so that acceleration produced by changing the move direction in circular interpolation does not exceed the maximum allowable acceleration rate specified in parameter No. 1735.
If the radius of an arc is very small, a calculated feedrate may become too low.
In such a case, the feedrate is prevented from decreasing below the value specified in this parameter.

1735	Maximum allowable acceleration rate for the deceleration function based on acceleration in circular interpolation for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Set a maximum allowable acceleration rate for the deceleration function based on acceleration in circular interpolation.
Feedrate is controlled so that acceleration produced by changing the move direction in circular interpolation does not exceed the value specified in this parameter.
For an axis with 0 set in this parameter, the deceleration function based on acceleration is disabled.
If a different value is set in this parameter for each axis, a feedrate is determined from the smaller of the acceleration rates specified for the two circular axes.

1737	Maximum allowable acceleration rate for the deceleration function based on acceleration in AI contour control for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Set a maximum allowable acceleration rate produced by changing the tool move direction.
For an axis with 0 set in this parameter, the deceleration function based on acceleration is disabled. If 0 is set for all axes, the deceleration function based on acceleration is not performed.
In circular interpolation, however, the deceleration function based on feedrate control using acceleration in circular interpolation (parameter No. 1735) is enabled.

1738	Minimum allowable feedrate for the deceleration function based on acceleration in AI contour control

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
With the deceleration function based on acceleration in AI contour control, a feedrate most suitable for a desired figure is automatically calculated.
Depending on the figure, however, the calculated feedrate may become too low.
In such a case, the feedrate is prevented from decreasing below the value specified in this parameter.
If overriding using the deceleration function based on cutting load is enabled, a feedrate lower than the minimum allowable feedrate may be used.

1763	FL rate for acceleration/deceleration after cutting feed interpolation for each axis in the acceleration/deceleration before interpolation mode

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set a minimum allowable feedrate (FL rate) for acceleration/ deceleration after cutting feed interpolation in look-ahead acceleration/deceleration before interpolation as in AI contour control.

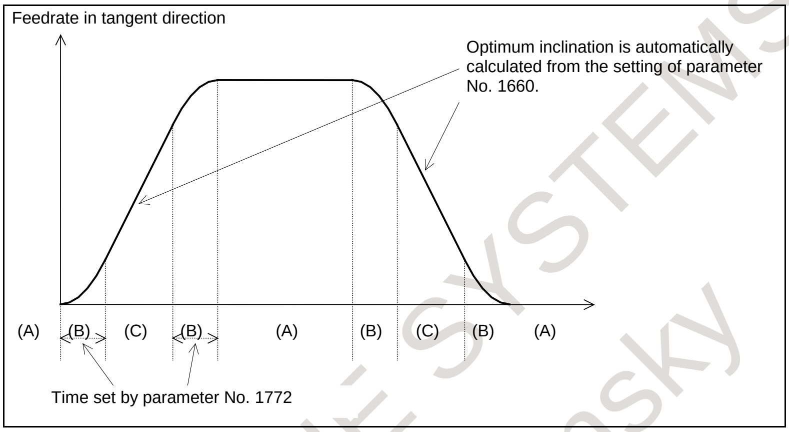
1769	Time constant for acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
In the look-ahead acceleration/deceleration before interpolation mode as in AI contour control, not the ordinary time constant (parameter No. 1622) but the value of this parameter is used.
Be sure to specify the same time constant value for all axes except for a special application. If different values are set, correct linear and circular figures cannot be obtained.

1772	Acceleration change time of bell-shaped acceleration/deceleration before interpolation

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 200

Set an acceleration change time of look-ahead bell-shaped acceleration/ deceleration before interpolation (time for changing from the state of constant feedrate (A) to the state of constant acceleration/deceleration (C) at the acceleration rate calculated from the acceleration rate set in parameter No. 1660: time of (B) in the figure).
If the setting is negative or 0, bell-shaped acceleration/deceleration before interpolation is invalid. If the setting is 200 or more, the setting is clamped by 200.



1783	Maximum allowable feedrate difference for feedrate determination based on corner feedrate difference

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

If a feedrate component change for each axis exceeding the value set in this parameter occurs at the joint of blocks, the feedrate determination function based on corner feedrate difference finds a feedrate not exceeding the set value and performs deceleration by using look-ahead acceleration/deceleration before interpolation. Thus, a shock to the machine and machining error at a corner can be reduced.
Feedrate determination function based on corner feedrate difference disabled for an axis this parameter is zero.

3241	Character blinking in the AI contour control I mode (first character)
to	to
3247	Character blinking in the AI contour control I mode (seventh character)

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 95

Set the first to seventh blinking characters in the AI contour control I mode by using ASCII codes represented as decimal numbers.
When 0 is set in all of these parameters, "AICC 1" blinks.
Code numbers 032 to 095 in the "CHARACTER CODE LIST" can be set.

3251	**Character blinking in the AI contour control II mode (first character)**
to	**to**
3257	**Character blinking in the AI contour control II mode (seventh character)**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 95

Set the first to seventh blinking characters in the AI contour control II mode by using ASCII codes represented as decimal numbers.
When 0 is set in all of these parameters, "AICC 2" blinks.
Code numbers 032 to 095 in the "CHARACTER CODE LIST" can be set.

CHARACTER CODE LIST

Character	Code	Comment	Character	Code	Comment	Character	Code	Comment
	032	Space	6	054		L	076	
!	033	Exclamation mark	7	055		M	077	
”	034	Quotation marks	8	056		N	078	
#	035	Sharp	9	057		O	079	
$	036	Dollar mark	:	058	Colon	P	080	
%	037	Percent	;	059	Semicolon	Q	081	
&	038	Ampersand	<	060	Left angle bracket	R	082	
’	039	Apostrophe	=	061	Equal sign	S	083	
(	040	Left parenthesis	>	062	Right angle bracket	T	084	
)	041	Right parenthesis	?	063	Question mark	U	085	
*	042	Asterisk	@	064	Commercial at mark	V	086	
+	043	Positive sign	A	065		W	087	
,	044	Comma	B	066		X	088	
-	045	Negative sign	C	067		Y	089	
.	046	Period	D	068		Z	090	
/	047	Slash	E	069		[	091	Left square bracket
0	048		F	070		¥	092	Yen mark
1	049		G	071		]	093	Right square bracket
2	050		H	072			094	
3	051		I	073		_	095	Underline
4	052		J	074				
5	053		K	075				

	#7	#6	#5	#4	#3	#2	#1	#0
7055					**BCG**			

[Input type] Parameter input
[Data type] Bit path

#3 BCG The pre-interpolation bell-shaped acceleration/deceleration time constant change function is:
0: Disabled.
1: Enabled.

NOTE
When function for changing time constant of look-ahead bell-shaped acceleration/deceleration before interpolation is enabled (bit 3 (BCG) of parameter No. 7055 is set to 1), be sure to specify acceleration rate of the reference axis (parameter No. 1660).
If acceleration rate of the reference axis is not specified, optimum acceleration rate is not obtained.

7066	Acceleration/deceleration reference speed for the time constant change function of bell-shaped acceleration/deceleration before interpolation

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter is used when the time constant change function of look-ahead bell-shaped acceleration/deceleration before interpolation is used.
If the setting is 0, bell-shaped acceleration/deceleration before interpolation is invalid.

	#7	#6	#5	#4	#3	#2	#1	#0
8451	NOF			ZAG				

[Input type] Setting input
[Data type] Bit path

#4 ZAG The deceleration function based on cutting load in AI contour control II (deceleration based on Z-axis fall angle) is:
0: Not performed.
1: Performed.
When this parameter is set to 1, be sure to set parameters Nos. 8456, 8457, and 8458.

#7 NOF In AI contour control II, an F command is:
0: Not ignored.
1: Ignored.
When this parameter is set to 1, the specification of the maximum allowable feedrate set in parameter No. 8465 is assumed.

8456	Override for area 2 that is applied during deceleration according to the cutting load in AI contour control II
8457	Override for area 3 that is applied during deceleration according to the cutting load in AI contour control II
8458	Override for area 4 that is applied during deceleration according to the cutting load in AI contour control II

[Input type] Setting input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 100

For the function of decelerating according to the cutting load in AI contour control II, the override set in a parameter can be applied according to the angle at which the tool moves downward along the Z-axis. The feedrate obtained according to other conditions is multiplied by the override for the area containing angle θ at which the tool moves downward.
However, when bit 1 (ZG2) of parameter No. 19515 is set to 0, no parameter is available to area 1, and 100% is applied at all times. When bit 1 (ZG2) of parameter No. 19515 is set to 1, set an override value for area 1 in parameter No. 19516.

Area 1 $0° \leq \theta < 30°$
Area 2 $30° \leq \theta < 45°$
Area 3 $45° \leq \theta < 60°$
Area 4 $60° \leq \theta \leq 90°$

	#7	#6	#5	#4	#3	#2	#1	#0
8459					OVRB			

[Input type] Parameter input
[Data type] Bit path

#3 OVRB For deceleration based on a feedrate difference or acceleration rate in AI contour control, override is:
0: Disabled.
1: Enabled.
Usually, override is enabled for a specified feedrate, and AI contour control is applied to the specified feedrate. When this parameter is set to 1, override is applied to a feedrate placed under AI contour control.

8465	Maximum allowable feedrate for AI contour control

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the maximum allowable feedrate for contour control.
If a feedrate higher than the setting of this parameter is specified in the AI contour control mode, the feedrate is clamped to that set in this parameter.
If this parameter is set to 0, no clamping is performed.
When bit 7 (NOF) of parameter No. 8451 is set to 1, the tool moves, assuming that the feedrate set in this parameter is specified. If 0 is set in this parameter at this time, a movement is made at the specified feedrate.

8466	Maximum allowable feedrate for AI contour control (when a rotation axis is singly specified)

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the maximum allowable feedrate for AI contour control when a rotation axis is singly specified.

If a feedrate higher than the setting of this parameter is specified in the AI contour control mode, the feedrate is clamped to that set in this parameter.
If this parameter is set to 0, the feedrate is clamped to that set in parameter No. 8465.
When bit 7 (NOF) of parameter No. 8451 is set to 1 and a rotation axis is singly specified, the tool moves, assuming that the feedrate set in this parameter is specified. If 0 is set in this parameter at this time, the tool moves at the feedrate specified in parameter No. 8465.

	#7	#6	#5	#4	#3	#2	#1	#0
11223				CML				

[Input type] Parameter input
[Data type] Bit path

#4 CML When AI contour control is effective and machine lock is applied:
0: Operates in accordance with conventional specifications.
1: Operates at almost the same speed and cycle time as machine lock is invalid.

	#7	#6	#5	#4	#3	#2	#1	#0
11240							AMP	

[Input type] Parameter input
[Data type] Bit path

#1 AMP Movement with G53 command, and movement from middle point to reference position with G28 or G30 command are done with:
0: Acceleration/deceleration after interpolation.
1: Acceleration/deceleration before interpolation.

NOTE
This parameter is enabled when the following conditions.
- In AI contour control mode
- Bit 4(ZRL) of parameter No.1015 is set to 1.
- Bit 1(LRP) of parameter No.1401 is set to 1
- Parameter No.1671 of at least one of axes is set to value except 0.
- Bit 5(FRP) of parameter No.19501 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
11549		AP5						

[Input type] Parameter input
[Data type] Bit

#6 AP5 In multi-path system, AI contour control is:
0: Not executed simultaneously by more than 5 paths. (Set to 0 necessarily if AI contour control is not executed simultaneously by more than 5 paths.)
1: Executed simultaneously by more than 5 paths. (Set to 1 necessarily if AI contour control is executed simultaneously by more than 5 paths.)

NOTE
1 When this parameter is set, the power must be turned off before operation is continued.
2 If AI contour control is executed simultaneously by more than 5 paths without setting this parameter to 1, sufficient performance might not be obtained.

	#7	#6	#5	#4	#3	#2	#1	#0
19500	FCC	FNW						

[Input type] Parameter input
[Data type] Bit path

#6 FNW When the feedrate is determined according to the feedrate difference and acceleration in AI contour control:
0: The maximum feedrate at which the allowable feedrate difference and acceleration for each axis are not exceeded is used.
1: A feedrate is determined to ensure that a constant deceleration rate is applied to the same figure regardless of the direction of movement.

#7 FCC When there is an axis that requires one or more seconds for acceleration in look-ahead acceleration/deceleration before interpolation:
0: Emphasis is placed on precision, so that the specified feedrate may not be reached.
1: Emphasis is placed on speed, so that the specified feedrate is produced.
When this parameter is set to 1, the precision of curved interpolation such as circular interpolation may decrease.

	#7	#6	#5	#4	#3	#2	#1	#0
19503				ZOL				HPF

[Input type] Parameter input
[Data type] Bit path

#0 HPF When a feedrate is determined based on acceleration in AI contour control II, smooth feedrate control is:
0: Not used.
1: Used.

#4 ZOL The deceleration function based on cutting load in AI contour control II (deceleration based on Z-axis fall angle) is:
0: Enabled for all commands.
1: Enabled for linear interpolation commands only.

	#7	#6	#5	#4	#3	#2	#1	#0
19515							ZG2	

[Input type] Parameter input
[Data type] Bit path

#1 ZG2 When the deceleration function based on cutting load in AI contour control II (deceleration based on Z-axis fall angle) is used:
0: Stepwise override values are applied.
1: Inclined override values are applied.
This parameter is valid only when bit 4 (ZAG) of parameter No. 8451 is set to 1.

When this parameter is set to 1, be sure to set parameters Nos. 19516, 8456, 8457, and 8458.

19516	Override for area 1 in deceleration based on cutting load in AI contour control II

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 100

This parameter sets an override value for area 1 when the deceleration function based on cutting load in AI contour control II is used.
This parameter is valid only when bit 1 (ZG2) of parameter No. 19515 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
19517							HNG	SNG

[Input type] Parameter input
[Data type] Bit path

#0 SNG When smooth speed control is effective, by block length of the linear interpolation, smooth speed control and speed control with change of acceleration on each axis;
0: Are not invalidated.
1: Are invalidated.
When smooth speed control is effective, smooth speed control and speed control with change of acceleration on each axis are invalidated in the block longer than the block length set to parameter No. 19518 if this parameter is set to 1.

#1 HNG By block length of the linear interpolation, speed control with acceleration on each axis and speed control with change of acceleration on each axis;
0: Are not invalidated.
1: Are invalidated.
Speed control with acceleration on each axis and speed control with change of acceleration on each axis are invalidated in the block longer than the block length set to parameter No. 19518 if this parameter is set to 1.

19518	Block length in speed control with acceleration or smooth speed control and speed control with change of acceleration are invalidated

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the block length to invalidate speed control with acceleration on each axis or smooth speed control and speed control with change of acceleration on each axis by block length of the linear interpolation.
This parameter is effective when bit 1 (HNG) of parameter No. 19517 is set to 1 or bit 0 (SNG) of parameter No. 19517 is set to 1 if smooth speed control is effective.
Speed control with acceleration on each axis of smooth speed control and speed control with change of acceleration on each axis are invalidated in the block longer than the block length set to this parameter.
If 0.0 is set, the specification of 10.0 is assumed.

Alarm and message

Number	Message	Description
PS5009	PARAMETER ZERO (DRY RUN)	The dry run rate parameter No. 1410 or the parameter for the maximum cutting feedrate for each axis is 0. The parameter for the maximum cutting feedrate for each axis is No. 1432 if look-ahead acceleration/deceleration before interpolation is enabled and No. 1430 otherwise. Functions that cause look-ahead acceleration/deceleration before interpolation include AI contour control.
PS5011	PARAMETER ZERO (CUT MAX)	The setting of the parameter for the maximum cutting feedrate is 0. The parameter is No. 1432 if look-ahead acceleration/deceleration before interpolation is enabled and No. 1430 otherwise. Functions that cause look-ahead acceleration/deceleration before interpolation include AI contour control.
PS5110	IMPROPER G-CODE (AICC MODE)	An unspecifiable G code was specified in the AI contour control mode.

Notes

- About processing macro statements

In AI contour control mode, the NC statements of multiple blocks are read ahead. Macro statements such as arithmetic expressions and conditional branches are processed as soon as they are buffered (i.e. read into the buffer). Therefore, the timing of the macro statement execution is not always the specified order.
In case that you need to execute the macro statement after completing the NC block just before the macro statement, specify M code or G code that is not buffered just before the macro statement. Specially, in case of reading/writing the system variables to control signals, coordinates, compensation value, etc., it may be different system variable data by the timing of the NC statement execution. To avoid this phenomenon, specify such M codes or G codes before the macro statement, if necessary.

- Setting of acceleration/deceleration, and feedrate

In the mode of look-ahead acceleration/deceleration before interpolation, when a time required for acceleration or deceleration is over one second, the feedrate could not reach the specified data.
When bit 7 (FCC) of parameter No.19500 is set to 1, the feedrate can reach the specified data.
In this case, however, the interpolation precision may be lower in curve interpolation such as circular interpolation.

7.1.12 Speed Command Extension in Least Input Increments C

Overview

When the least input increment C (IS-C) is selected, the limitations indicated in Table 7.1.12 (a) and Table 7.1.13 (b) have conventionally been applied to the speed and acceleration parameters.
For example, when the unit of data is mm, and the increment system is IS-C, the maximum rapid traverse rate and maximum cutting speed have been unable to exceed 100000 mm/min due to the limitations on the valid data range.

- Conventional speed and angular velocity parameters

Table 7.1.12 (a)

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min deg/min	IS-C	0.0001	0.0000 to +99999.9999
inch/min	IS-C	0.00001	0.00000 to +4000.00000

- Conventional acceleration and angular acceleration parameters

Table 7.1.12 (v)

Unit of data	Increment system	Minimum data unit	Valid data range
mm/sec^2 deg/sec^2	IS-C	0.0001	0.0000 to +99999.9999
inch/sec^2	IS-C	0.00001	0.00000 to +9999.99999

With an axis for which bit 7 (IESP) of parameter No. 1013 is set to 1 and IS-C is specified as the increment system, this function extends the parameter input limitations to those indicated in Table 7.1.13 (c) and Table 7.1.13 (d).
With this function, values greater than the conventionally allowed values can be set in the speed and acceleration parameters.
For example, the upper limit of the maximum rapid traverse rate and maximum cutting speed after this function is applied is:
- About 1000000 mm/min when the unit of data is mm, and IS-C is used

- Speed and angular velocity parameters when this function is used

Table 7.1.13 (c)

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min deg/min	IS-C	0.001	0.000 to +999000.000
inch/min	IS-C	0.0001	0.0000 to +9600.0000

- Acceleration and angular acceleration parameters when this function is used

Table 7.1.13 (d)

Unit of data	Increment system	Minimum data unit	Valid data range
mm/sec^2 deg/sec^2	IS-C	0.001	0.000 to +999999.999
inch/sec^2	IS-C	0.0001	0.0000 to +99999.9999

> **NOTE**
> Note that the least input increment and valid data range of set data vary from the conventional ones.

- Feed forward

In case that the speed command is extended, when the feed forward function is enabled (bit 1 (FEED) of parameter No.2005 = 1), please set to 1 bit 3 (ISE64) of parameter No.2282.

Limitation

- Time constant parameters for linear acceleration/deceleration after interpolation and bell-shaped acceleration/deceleration after interpolation

If linear acceleration/deceleration after interpolation or bell-shaped acceleration/deceleration after interpolation is used in an operation such as rapid traverse, cutting feed, or manual feed, the maximum allowable time constant value is a half of the conventional maximum allowable value.
The table below indicates the pertinent time constant parameters:

Parameter number	Meaning
1620	Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis
1621	Time constant T2 used for bell-shaped acceleration/deceleration in rapid traverse for each axis
1622	Time constant of acceleration/deceleration in cutting feed for each axis
1624	Time constant of acceleration/deceleration in jog feed for each axis
1626	Acceleration/deceleration time constant in threading cycles for each axis

Parameter number	Meaning
1769	Time constant for acceleration/deceleration after cutting feed interpolation in the look-ahead acceleration/deceleration before interpolation mode
5271-5274	Time constant for acceleration/deceleration in rigid tapping extraction (first-stage to fourth-stage gear)
5365-5368	Bell-shaped acceleration/deceleration time constant in rigid tapping (first-stage to fourth-stage gear)

- VCMD waveform display function

As the speed increases, the amount of data that can be obtained for VCMD waveform display increases. So, the maximum treatable amount of data may be exceeded, resulting in incorrect waveform display.

- Positional deviation amount display on the servo setting screen

If the positional deviation amount exceeds -9999999 when a fast movement is made in the negative direction, the displayed number of digits decreases by one digit due to a limitation on display space.
To reference a correct positional deviation amount, check diagnosis data No. 300, "Servo positional deviation".

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1013	IESPx							

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#7 IESPx When the least input increment is C (IS-C), the function to allow to set the larger value to the parameter of the speed and the acceleration:
0: Not used.
1: Used.

As for the axis which set this parameter when the least input increment is C (IS-C), the larger value can be set to the parameter of the speed and the acceleration.
The valid data ranges for the speed and acceleration parameters are indicated in Table 7.1.13 (c) and Table 7.1.13 (d) provided in "Overview".
When this function is made effective, the digit number below the decimal point of the parameter on input screen is changed. The digit number below the decimal point decreases by one digit in case of the least input increment C (IS-C).

	#7	#6	#5	#4	#3	#2	#1	#0
2282					ISE64			

[Input type] Parameter input
[Data type] Bit axis

#3 ISE64 The speed limitation on the feed forward function (bit 1 (FEED) of parameter No. 2005 = 1) is:
0: Remains unchanged from the conventional value.
1: Extended.
When the feed forward function is enabled, the speed limitation on an axis for which this parameter is set is extended as indicated below if the increment system is IS-C.

	IS-C	
	Conventional	Extended
Speed [Unit: mm/min]	100000	999000

Notes

NOTE
1 To use this function, the corresponding servo software is required.
2 This function cannot be set for an axis under PMC axis control.

7.2 ACCELERATION/DECELERATION CONTROL

7.2.1 Automatic Acceleration/Deceleration

7.2.1.1 Automatic acceleration/deceleration

Overview

To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement (Fig. 7.2.1.1 (a)).

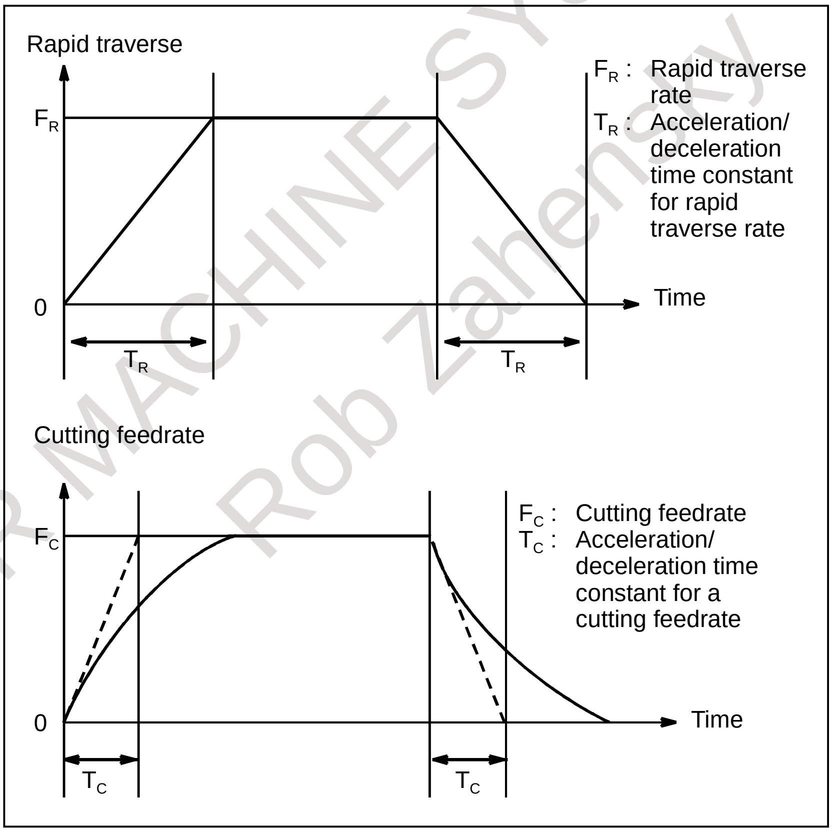
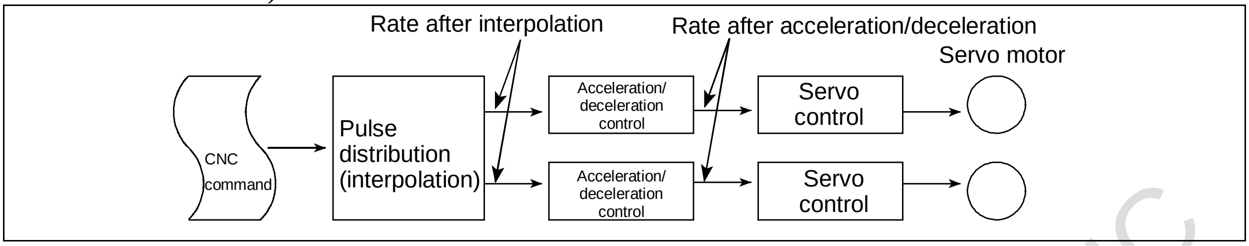


Fig. 7.2.1.1 (a)

Acceleration and deceleration is performed when starting and ending movement, resulting in smooth start and stop.
Automatic acceleration/deceleration is also performed when feedrate changes, so the change in speed is also smooth.
It is not necessary to take acceleration/deceleration into consideration when programming.
Rapid traverse : Linear acceleration/deceleration (time constant per axis is set by parameter No. 1620)
Cutting feed : Exponential acceleration/deceleration (time constant per axis is set by parameter 1622)

Jog feed : Exponential acceleration/deceleration (time constant per axis is set by parameter No. 1624)



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1610			**THLx**	**JGLx**			**CTBx**	**CTLx**

[Input type] Parameter input
[Data type] Bit axis

#0 CTLx Acceleration/deceleration in cutting feed or dry run during cutting feed
0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied.

> **NOTE**
> This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

#1 CTBx Acceleration/deceleration in cutting feed or dry run during cutting feed
0: Exponential acceleration/deceleration or linear acceleration/ deceleration is applied. (depending on the setting in bit 0 (CTLx) of parameter No. 1610)
1: Bell-shaped acceleration/deceleration is applied.

> **NOTE**
> This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

#4 JGLx Acceleration/deceleration in jog feed
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied. (Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)

> **NOTE**
> This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

#5 THLx Acceleration/deceleration in threading cycles
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied. (Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610) As the time constant and FL rate, however, the settings of parameters Nos. 1626 and 1627 for threading cycles are used.

1620	Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Specify a time constant used for acceleration/deceleration in rapid traverse.

[Example]

For linear acceleration/deceleration

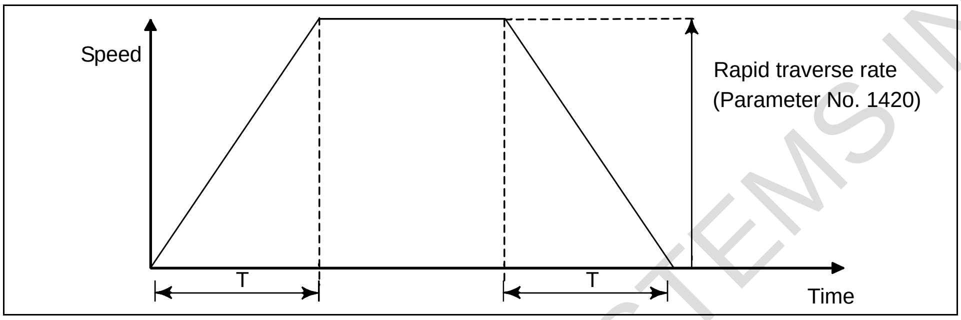


Fig. 7.2.1.1 (b)

T : Setting of parameter No. 1620

For bell-shaped acceleration/deceleration

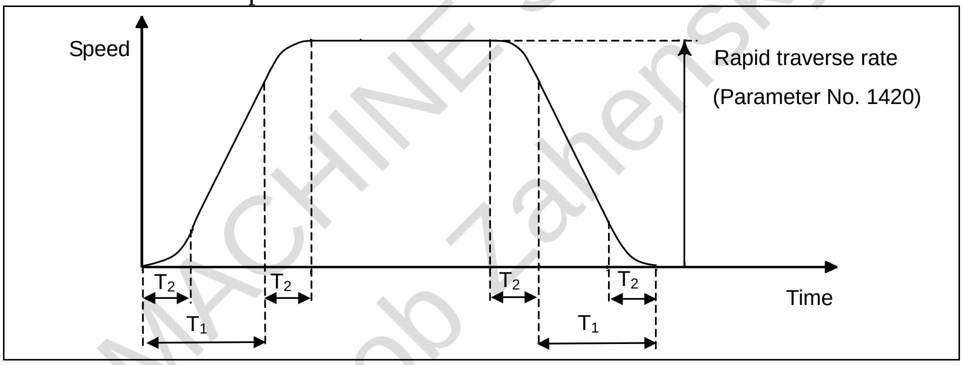


Fig. 7.2.1.1 (c)

T_1 : Setting of parameter No. 1620
T_2 : Setting of parameter No. 1621
(However, $T_1 \geq T_2$ must be satisfied.)
Total acceleration (deceleration) time : $T_1 + T_2$
Time for linear portion : $T_1 - T_2$
Time for curve portion : $T_2 \times 2$

1621	Time constant T_2 used for bell-shaped acceleration/deceleration in rapid traverse for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 512

Specify time constant T_2 used for bell-shaped acceleration/ deceleration in rapid traverse for each axis.

1622	Time constant of acceleration/deceleration in cutting feed for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in cutting feed, bell-shaped acceleration/deceleration after interpolation or linear acceleration/ deceleration after interpolation in cutting feed for each axis. Which type to use is selected with bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610. Except for special applications, the same time constant must be set for all axes in this parameter. If the time constants set for the axes differ from each other, proper straight lines and arcs cannot be obtained.

1623	FL rate of exponential acceleration/deceleration in cutting feed for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set the lower limit (FL rate) of exponential acceleration/deceleration in cutting feed for each axis.

1624	Time constant of acceleration/deceleration in jog feed for each axis.

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set the time constant used for acceleration/deceleration in jog feed for each axis.

1625	FL rate of exponential acceleration/deceleration in jog feed for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set the FL rate of exponential acceleration/deceleration in jog feed for each axis.
This parameter allows only the exponential type.

1626	Acceleration/deceleration time constant in threading cycles for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set a time constant for acceleration/deceleration after interpolation in the threading cycles G92 (T series), G76 (T series), and G76.7 (M series) for each axis.

1627	FL rate for acceleration/deceleration in threading cycles for each axis

[Input type] Parameter input

[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set an FL rate for acceleration/deceleration after interpolation in the threading cycles G92 (T series), G76 (T series), and G76.7 (M series) for each axis. Set 0 at all times except in a special case.

7.2.1.2 Rapid traverse block overlap

Overview

If rapid traverse blocks continue or the block next to a rapid traverse block does not move, the next block can be executed when the feedrate of each axis of a rapid traverse block is decreased to the reduction ratio set by the parameter.

Explanation

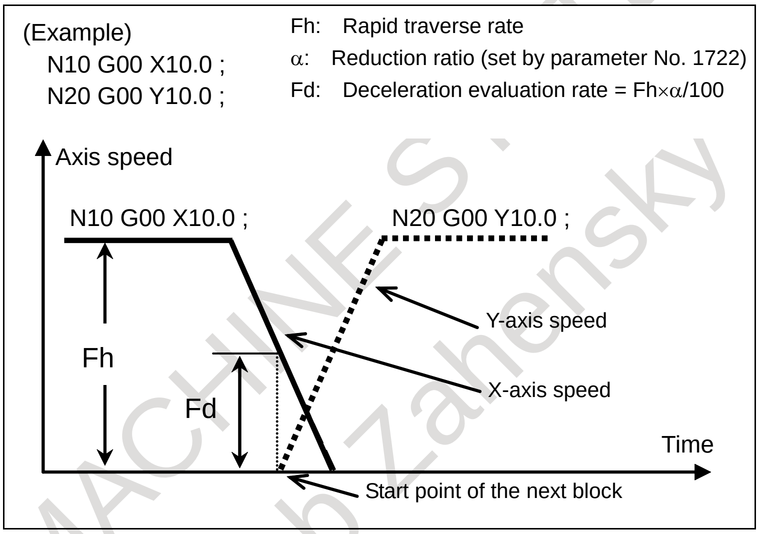


Fig. 7.2.1.2

> **NOTE**
> Whether the speed is reduced to the reduction ratio is determined based on parameter No. 1420 (rapid traverse rate for each axis).

Limitation

Rapid traverse block overlap is disabled when:

- Pre-interpolation acceleration/deceleration is used for rapid traverse.
- The rapid traverse block overlap disable signal ROVLP <Gn053.5> is "1".
- In the rapid traverse command, dry run is enabled (bit 6 (RDR) of parameter No. 1401 = 1) and dry run signal DRN <Gn046.7> is "1". (This function is not disabled, however, if manual rapid traverse selection signal RT <Gn019.7> is set to "1".)
- Rapid traverse is of linear interpolation type (bit 1 (LRP) of parameter No. 1401 = 1) and the acceleration/deceleration is constant over time (bit 4 (PRT) of parameter No. 1603 = 1).

Signal

Rapid traverse block overlap disable signal ROVLP<Gn053.5>

[Classification] Input signal
[Function] This signal disables rapid traverse block overlap.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn053			ROVLP					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601				RTO				

[Input type] Parameter input
[Data type] Bit path

#4 RTO Block overlap in rapid traverse
0: Blocks are not overlapped in rapid traverse.
1: Blocks are overlapped in rapid traverse.

1722	Rapid traverse feedrate reduction ratio for overlapping rapid traverse blocks

[Input type] Parameter input
[Data type] Byte axis
[Unit of data] %
[Valid data range] 0 to 100

This parameter is used when rapid traverse blocks are arranged successively, or when a rapid traverse block is followed by a block that does not cause movement. When the feedrate for each axis of a block is reduced to the ratio set in this parameter, the execution of the next block is started.

NOTE
The parameter No. 1722 is effective when bit 4 (RTO) of parameter No. 1601 is set to 1.

7.2.1.3 Programmable rapid traverse overlap

Overview

The programmable rapid traverse overlap supports:
- Changing the feedrate reduction ratio for rapid traverse overlap from the macro program becomes possible.
- Shortening the cycle time becomes possible by doing rapid traverse overlap between threading cycle blocks.

- Programmable rapid traverse overlap

If the bit 0 (PRO) of parameter No. 1612=0, the feedrate reduction ratio for rapid traverse overlap is specified by the parameter No. 1722. If the parameter PRO=1, the ratio is specified by the system variables. Even if the parameter PRO=1, and the system variables other than 1 to 100 is specified, the ratio is specified by the parameter No. 1722.

#100851 Feedrate reduction ratio for rapid traverse overlap (1st axis)
#100852 Feedrate reduction ratio for rapid traverse overlap (2nd axis)
:
#100874 Feedrate reduction ratio for rapid traverse overlap (24th axis)

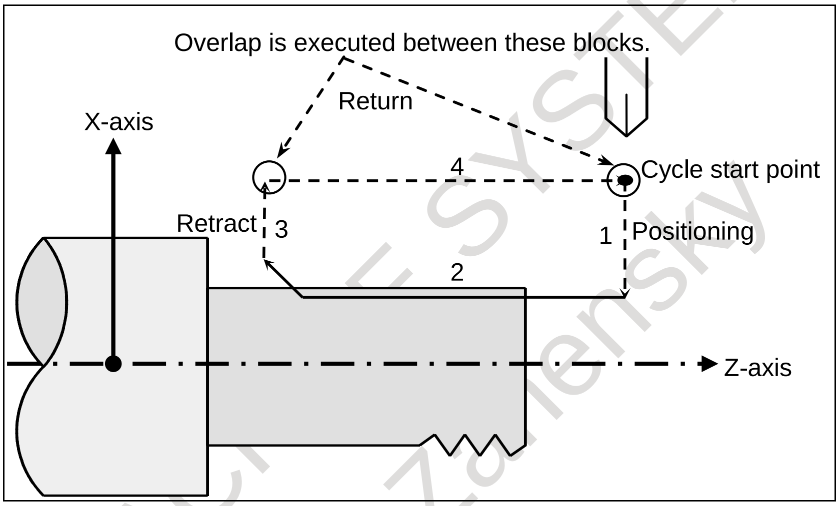
> **NOTE**
> 1 Available up to the number of controlled axes.
> 2 After the value is set, it is available until the system is reset.

To use this function, set the following signals and parameters.

- ROVLP<Gn053.5> is "0"
  Rapid traverse block overlap disable signal
- Bit 4 (RTO) of parameter No. 1601=1
  Blocks are overlapped in rapid traverse.
- Bit 0 (PRO) of parameter No. 1612=1
  The feedrate reduction ratio for rapid traverse overlap is specified by the system variables.

- Rapid traverse overlap between threading cycle blocks

Rapid traverse overlap is executed between "retract(3)" and "return(4)" and between "return(4)" and "next-threading cycle positioning(1)".



To use this function, set the following parameters.

- Bit 0 (CFR) of parameter No. 1611=1 :
  For retraction after threading, the type of acceleration /deceleration after interpolation for rapid traverse is used.
- Bit 4 (PRT) of parameter No. 1603=0 :
  For positioning, acceleration/deceleration of acceleration fixed type is used.

The thread cutting cycle retract function is disabled.

Explanation

- Programmable rapid traverse overlap

How to specify the feedrate reduction ratio for rapid traverse overlap by the system variables.

Example)

O0001;
N10 #100851=50;　　Write to system variable (ratio is 50%)
N20 G00 X10.00;
N30 G00 Z10.00;　　} Execute overlap (ratio is 50%)
;
N50 #100851=100;　　Rewrite to system variable (ratio is 100%)
N60 G00 X30.00;
N70 G00 Z50.00;　　} Execute overlap (ratio is 100%)
N80 M02;

> **NOTE**
> 1 To specify the feedrate reduction ratio for rapid traverse overlap by the system variables is executed as macro program.
> 2 Whether the feedrate has been reduced as specified by the reduction ratio is compare with the parameter No. 1420 (Rapid traverse rate for each axis).

- Rapid traverse overlap between threading cycle blocks

To use rapid traverse overlap between threading cycle blocks, set the bit 3 (TCO) of parameter No. 1611 = 1.

Rapid traverse overlap between threading cycle blocks is executed as shown Fig. 7.2.1.3.

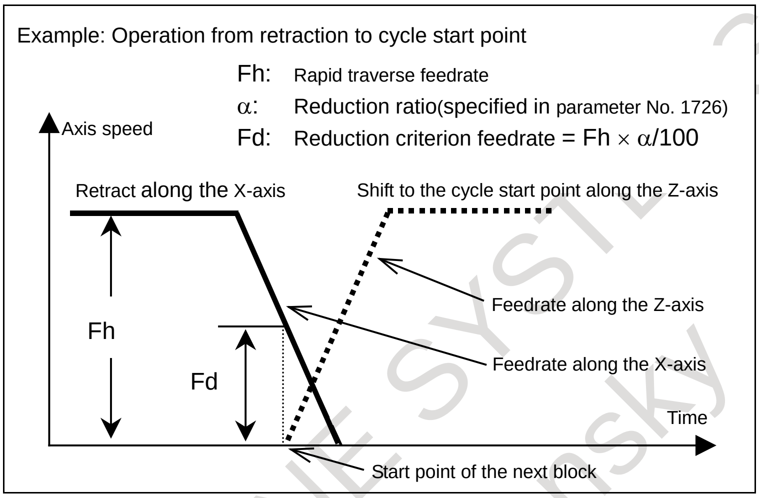


Fig. 7.2.1.3

> **NOTE**
> Whether the feedrate has been reduced as specified by the reduction ratio is compare with the parameter No. 1420 (Rapid traverse rate for each axis).

Restrictions

In following conditions, both rapid traverse block overlap and rapid traverse overlap between threading cycle blocks are disabled.

- Look-ahead acceleration/deceleration before interpolation is used in rapid traverse
- The bit 6 (RDR) of parameter No. 1401 = 1 (dry run for rapid traverse is enabled), and the dry run signal DRN<Gn046.7> is "1" and the manual rapid traverse selection signal RT <Gn019.7> is set to "0".
- The bit 1 (LRP) of parameter No. 1401 = 1 (positioning is performed with linear interpolation) and bit 4 (PRT) of parameter No. 1603=1 (acceleration/deceleration of time fixed type is used).

・Restrictions about programmable rapid traverse overlap

- Rapid traverse block overlap is disabled if the rapid traverse block overlap disable signal ROVLP<Gn053.5> is "1".

・Restrictions about rapid traverse overlap between threading cycle blocks

- If this function is enabled, thread cycle retract function is disabled.
- The threading cycle command must be commanded in G40 mode (Cutter/tool nose radius compensation cancel mode).
- Rapid traverse block overlap disable signal ROVLP<Gn053.5> is invalid.

Signal

Rapid traverse block overlap disable signal ROVLP<Gn053.5>

[Classification] Input signal

[Function] This signal disables rapid traverse block overlap.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn053			ROVLP					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601				RTO				

[Input type] Parameter input

[Data type] Bit path

#4 RTO Block overlap in rapid traverse
0: Blocks are not overlapped in rapid traverse.
1: Blocks are overlapped in rapid traverse.

	#7	#6	#5	#4	#3	#2	#1	#0
1611					TCO			

[Input type] Parameter input

[Data type] Bit path

#3 TCO Blocks in a threading cycle are:
0: Not overlapped in rapid traverse.
1: Overlapped in rapid traverse.

In a threading cycle that is a canned cycle, the rapid traverse overlap function can perform rapid traverse overlap between a retract operation and return operation and between a return operation and a positioning operation for the next threading cycle.
The rapid traverse deceleration ratio in an overlap between blocks is set in parameter No. 1726.

> **NOTE**
> When this function is used, threading cycle retraction is disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
1612								PRO

[Input type] Parameter input

[Data type] Bit path

#0 PRO The feedrate reduction ratio for rapid traverse overlap is specified by :
0: The parameter No. 1722
1: The system variables #100851[#_ROVLP[1]]-#100874[#_ROVLP[24]]

This function is available in following conditions.
Rapid traverse block overlap disable signal ROVLP<Gn053.5>="0" and bit 4 (RTO) of parameter No. 1601=1(Blocks are overlapped in rapid traverse).

1722	Rapid traverse feedrate reduction ratio for overlapping rapid traverse blocks

[Input type] Parameter input
[Data type] Byte axis
[Unit of data] %
[Valid data range] 0 to 100

This parameter is used when rapid traverse blocks are arranged successively, or when a rapid traverse block is followed by a block that does not cause movement. When the feedrate for each axis of a block is reduced to the ratio set in this parameter, the execution of the next block is started.

> **NOTE**
> The parameter No. 1722 is effective when bit 4 (RTO) of parameter No. 1601 is set to 1.

1726	Rapid traverse rate reduction ratio for overlapping threading cycle blocks

[Input type] Parameter input
[Data type] Byte axis
[Unit of data] %
[Valid data range] 0 to 100

In a threading cycle that is a canned cycle, when the feedrate for each axis is reduced to the deceleration ratio set in this parameter between a retraction and a return to the cycle start point or between a return to the cycle start point and a movement to the threading start point, the execution of the next block is started.

> **NOTE**
> The parameter No. 1726 is valid when bit 3 (TCO) of parameter No. 1611 is set to 1.

7.2.1.4 Smart overlap

Outline

Cycle time can be reduced by Smart overlap that can overlap at rapid traverse to cutting feed, cutting feed to rapid traverse, and rapid traverse to rapid traverse.

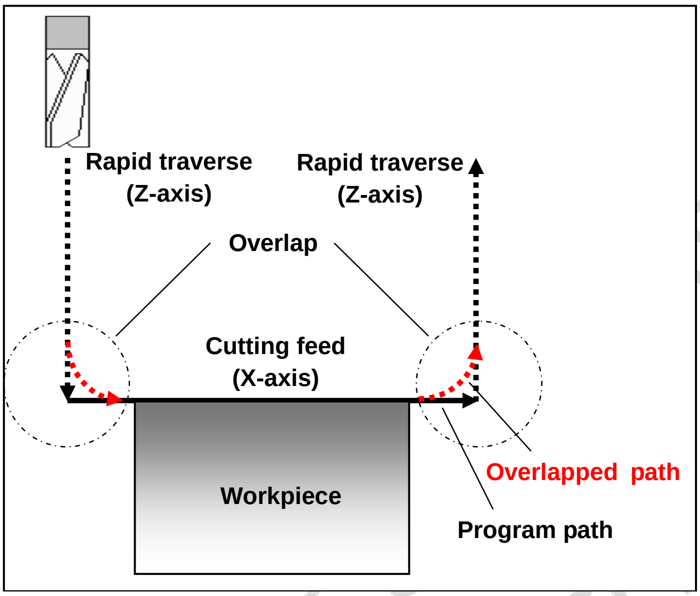


Fig. 7.2.1.4 (a) Overlapped path

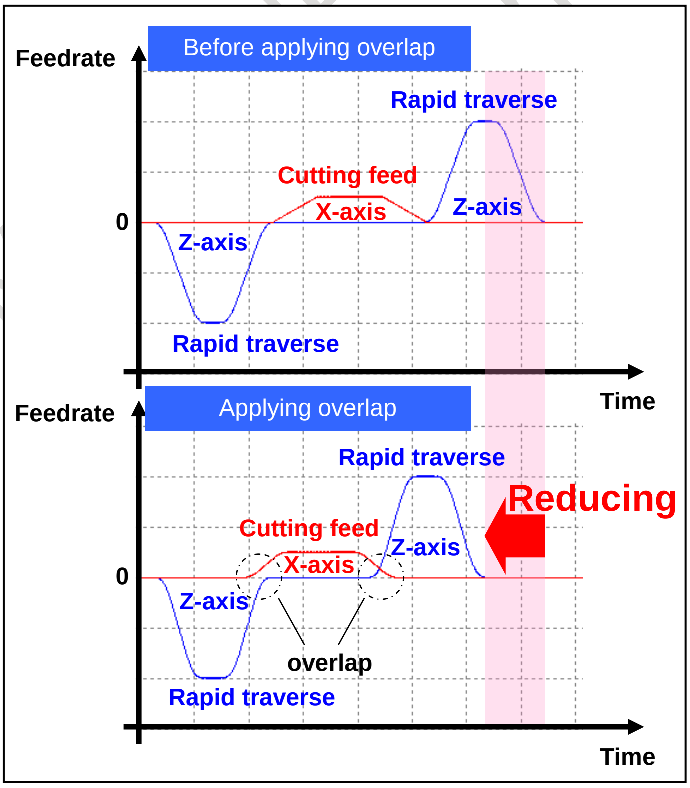


Fig. 7.2.1.4 (b) Effect of overlap

Explanation

When bit 1 (SOVx) of parameter (No.11236) is set to 1, Smart overlap is effective.
Overlap time can be set by parameter (Nos.11248 and 11249). The commands between blocks can be overlapped at the amount of the set overlap time. If the commands to which overlap is effective is consecutive, it overlaps between not only two blocks but two or more blocks.

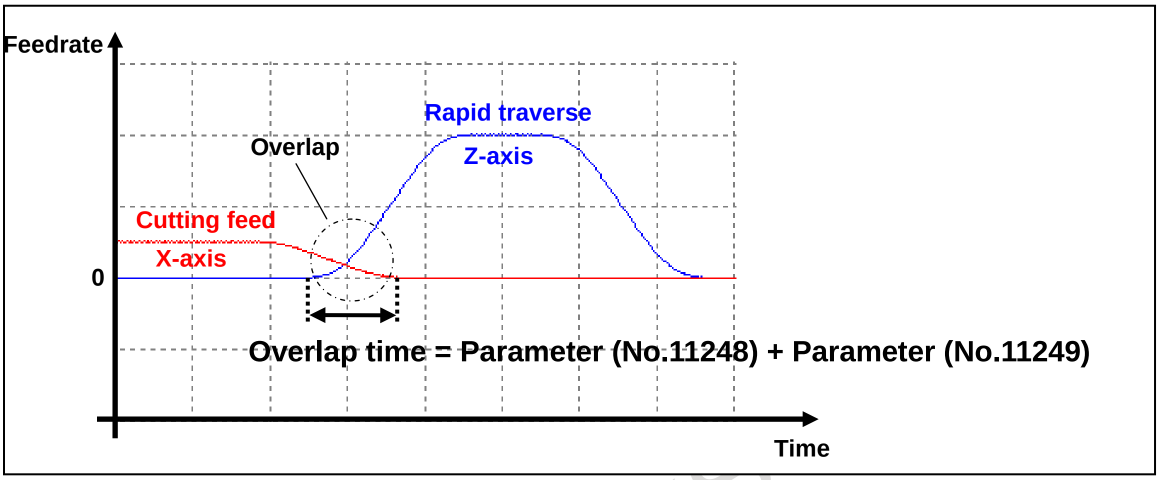


Fig.7.2.1.4 (c) Overlap time

･ Method for parameter setting

Set the parameters as follows to use Smart overlap is used.

1. For the axis for which Smart overlap is to be enabled, set bit 1 (SOVx) of parameter (No. 11236) to 1.

2. Set overlap time to parameter (No.11248).
   - Parameter (No.11248) is used as a common time constant for acceleration/deceleration after interpolation in rapid traverse and cutting feed. (This is called as acceleration/deceleration by Smart overlap.)
   - Usually set below the value of parameter (No.1622) (Time constant of acceleration/deceleration in cutting feed for each axis).
     In addition, set below the value of parameter (No.1769) (Time constant for acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode) when AICC (AI Contour Control) is used.
   - Set the same value to all axes to enable Smart overlap.
   - Usually set 0 to parameter (No.11249).

3. Set the following value to parameter (No.1621) (Time constant T2 used for acceleration/deceleration in rapid traverse for each axis).
   - Parameter (No.1621) = Parameter (No.1621) - Parameter (No.11248)
   - If the value in the above expression is minus, set 0 to parameter (No.1621).
   - If vibration is generated by the rapid traverse command, adjust the parameter (No.1621) of axes to which the vibration is generated by setting a bigger value.

4. Set the following value to parameter (No.1622) (Time constant of acceleration/deceleration in cutting feed for each axis).
   - Parameter (No.1622) = Parameter (No.1622) - Parameter (No.11248)
   - If the value in the above expression is minus, set 0 to parameter (No.1622).

5. When AICC (AI Contour Control) is used, set the following value parameter (No.176) (Time constant for acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode).
   - Parameter (No.1769) = Parameter (No.1769) - Parameter (No.11248)

- If the value in the above expression is minus, set 0 to parameter (No.1769).

6. When rapid traverse acceleration/deceleration before interpolation is used, set the following value parameter (No.11242) (Time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse).
   - Parameter (No.11242) = Parameter (No.11242) - Parameter (No.11248)
   - If the value in the above expression is minus, set 0 to parameter (No.11242).

   If parameter (No.11242) is 0, set the following value to parameter (No.1672) (Acceleration change time of bell-shaped acceleration/deceleration before interpolation for linear rapid traverse) because the adjustment by time constant for acceleration/deceleration after interpolation is unavailable.
   - Parameter (No.1672) = Parameter (No.1672) - Parameter (No.11248)
   - If the value in the above expression is minus, set 0 to parameter (No.1672).

7. When the each function of Table 7.2.1.4 (a), adjust time constants for acceleration/deceleration after interpolation of the each function.
   - The value of parameter (No.11248) is subtracted from the time constants of the each function.
   - When the bell-shaped acceleration/deceleration is “constant acceleration/deceleration type” + “constant acceleration/deceleration time type”, the value of parameter (No.11248) is subtracted from the time constant of “constant acceleration/deceleration time type”.
   - If the problem (vibrate is generated, and etc.) occurs on each function, adjust the time constants of the each function by setting a bigger value.

Table 7.2.1.4 (a) Time constants for acceleration/deceleration after interpolation of the each function

Function	Parameter
TIME CONSTANT FOR ACC/DEC IN JOG	No.1624
T CONSTANT AFTER IPL(THREAD CYCLE)	No.1626
TIME CNST2 AFTER IP(RAPID)	No.1674
TIME CNST2(BELL) AFTER IP(RAPID)	No.1675
TIME CNST3 AFTER IP(RAPID)	No.1676
TIME CNST3(BELL) AFTER IP(RAPID)	No.1677
TIME CNST2 AFTER IP(CUT)	No.1678
TIME CNST3 AFTER IP(CUT)	No.1679
TIME CONSTANT FOR ACC/DEC (GEAR 1 TO 4)	Nos.5261 to 5264
TIME CNST IN EXTRACTION (GEAR 1 TO 4)	Nos.5271 to 5274
TIME CNST FOR BELL (GEAR 1 TO 4)	Nos.5365 to 5368
LEVEL 1 TO 7 RAPID TRAVERSE TIME CONSTANT (Positioning by optimum accelerations)	Nos.6171 to 6177
TIME CNST T2 FOR LEVEL 1 TO 7 RAPID BELL (Positioning by optimum accelerations)	Nos.6191 to 6197
T CONSTANT AFTER IPL(SKIP FUNCTION)	No.6280
T CONSTANT AFTER RAPID (MH RETRACE)	No.6495
T CONST BELL AFTER RAPID(MH RETRACE)	No.6496
T CONST AFTER CUT (M-HANDLE RETRACE)	No.6497
T CONSTANT FOR LINEAR (RETRACT)	No.7745
T CONST FOR LINEAR IN RAPID(S-IMPOS)	No.8192
T CONST FOR BELL IN RAPID (S-IMPOSE)	No.8195
T CONST IN CUT (2 AXES SUPERIMPOSED)	No.8196
T CONST FOR LINEAR IN RAPID(3 S-IMP)	No.8197
T CONST FOR BELL IN RAPID (S-IMPOSE)	No.8198
T CONST IN CUT (3 AXES SUPERIMPOSED)	No.8199
TIME CONSTANT AFTER IPL(SP WITH SV)	No.11016
T CONST AFTER CUT IPL IN BEFOR IPL	No.11052
TIME CONST FOR RIGID TAPPING (G1 TO G4)	Nos.11060 to 11063
TIME CONST IN EXTRACT FOR R-TAP(G1 TO G4)	Nos.11065 to 11068
T-CON AIPL IN BIPL(RAPID)(msec)	No.11242

Function	Parameter
T CONST IN RAPID (TCP OR WSC MODE)	No.11263

> ⚠CAUTION
> If the overlap time is set bigger value than the original setting time constant for acceleration/deceleration after cutting feed interpolation, the machining accuracy of the workpiece might deteriorate.

[Example of parameter setting]

The example of parameter setting in case of setting overlap time as 16msec is shown below.

Table 7.2.1.4 (b) Example of parameter setting

Move command	Axis	Before applying Smart overlap				
		Parameter				Total (*1)
Rapid traverse		**No.1620**	**No.1621**	**-**	**No.11248**	-
	X	80	32	0		112
	Y	80	40	0		120
	Z	100	48	0		148
Cutting feed		**No.1622**	**-**	**-**		-
	X	32	0	0		32
	Y	32	0	0		32
	Z	32	0	0		32
Rapid traverse acc/dec before interpolation		**No.1671**	**No.1672**	**No.11242**	0	-
	X	8000	32	16		16
	Y	8000	40	16		16
	Z	6000	48	16		16
Cutting feed (AICC)		**No.1660**	**No.1772**	**No.1769**		-
	X	4000		16		16
	Y	4000	24	16		16
	Z	4000		16		16
Move command	**Axis**	**Applying Smart overlap**				
		Parameter				**Total (*1)**
Rapid traverse		**No.1620**	**No.1621**	**-**	**No.11248**	-
	X	80	16	0		112
	Y	80	24	0		120
	Z	100	32	0		148
Cutting feed		**No.1622**	**-**	**-**		-
	X	16	0	0		32
	Y	16	0	0		32
	Z	16	0	0		32
Rapid traverse acc/dec before interpolation		**No.1671**	**No.1672**	**No.11242**	16	-
	X	8000	32	0		16
	Y	8000	40	0		16
	Z	6000	48	0		16
Cutting feed (AICC)		**No.1660**	**No.1772**	**No.1769**		-
	X	4000		0		16
	Y	4000	24	0		16
	Z	4000		0		16

(*1) Total of time constant for acceleration/deceleration after interpolation (Parameter No.1660, No.1772, No.1671 and No.1672 are excluded for acceleration/deceleration before interpolation)

Parameter No.1620: Time constant T1 used for acceleration/deceleration after interpolation in rapid traverse
Parameter No.1660: Maximum allowable acceleration rate in acceleration/deceleration before interpolation (AICC)
Parameter No.1772: Acceleration change time of bell-shaped acceleration/deceleration before interpolation (AICC)
Parameter No.1671: Maximum allowable acceleration rate in acceleration/deceleration before interpolation for linear rapid traverse
Parameter No.1672: Acceleration change time of bell-shaped acceleration/deceleration before interpolation for linear rapid traverse

[Example]
This example is the case of automatic operation on parameter setting of Table 7.2.1.4 (b) without AICC.

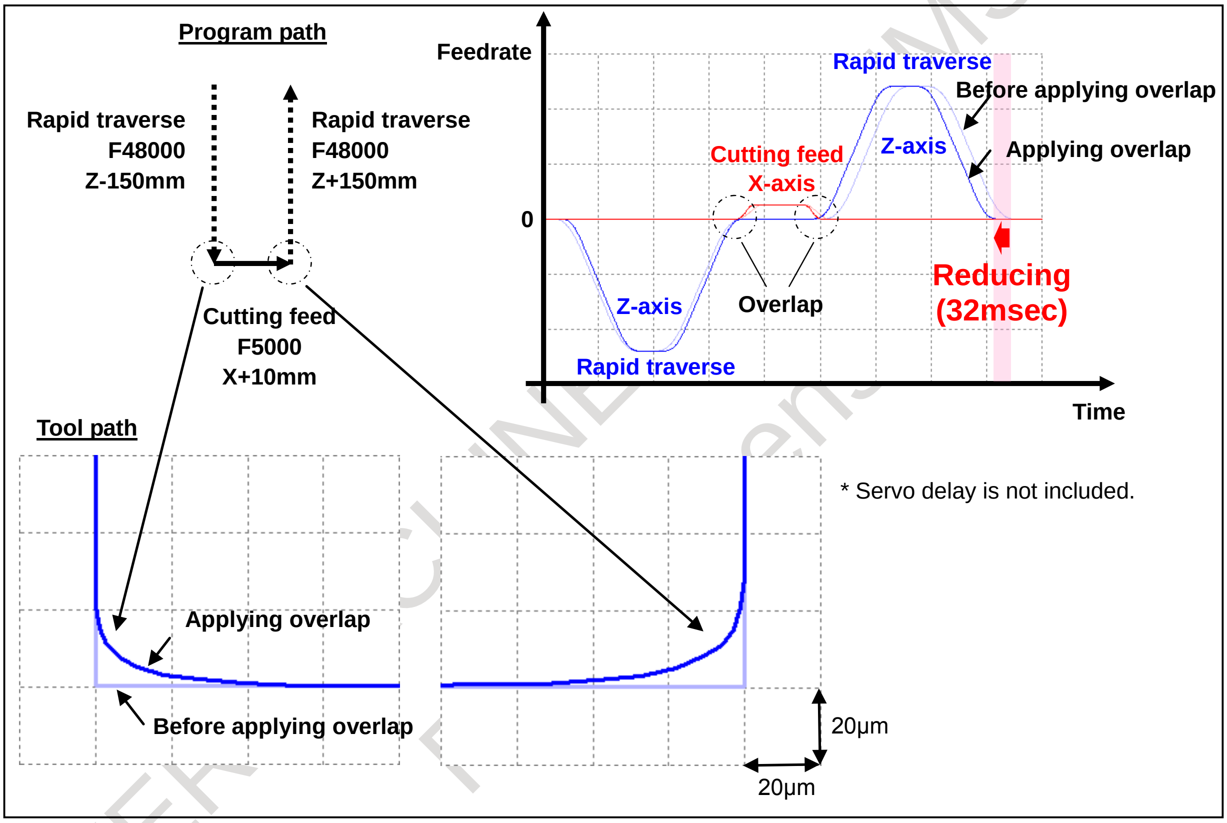


Fig. 7.2.1.4 (d) Example

- Clamp of overlap value

As the feedrate increases, overlap value becomes larger. To suppress it, by setting parameter (Nos.10740 to 10742), the overlap value can be clamped depending on program command.
(During Smart overlap, normal in-position check is not executed.)

Table 7.2.1.4 (c) Maximum overlap value

Program command		Maximum overlap value (Parameter Number)
Current block	Next block	
Rapid traverse	Rapid traverse	No.10740
Rapid traverse	Cutting feed	No.10741
Cutting feed	Rapid traverse	No.10742

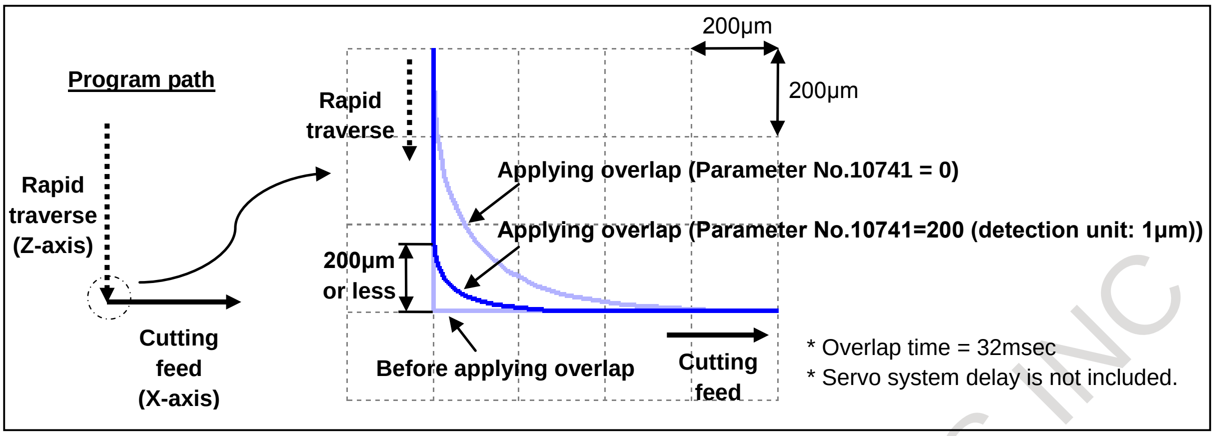


Fig. 7.2.1.4 (e) Clamp of overlap value

> **NOTE**
> 1 When parameter (Nos.10740 to 10742) is set to 0, the overlap value does not be clamped.
> 2 When you clamp overlap value, set the value of in-position width or more to parameter Nos.10740 to 10742.

- Disabling overlap

By setting 1 to bits 0 to 7 of parameter (No.11247), overlap can be disabled depending on program command.

Table 7.2.1.4 (d) Disabling overlap

Program command		Parameter		overlap
Current block	Next block	Number	value	
Rapid traverse	Rapid traverse	SRR(No.11247#0)	0	Enabled
			1	Disabled
	Cutting feed	SRC(No.11247#1)	0	Enabled
			1	Disabled
	No movement	SRN(No.11247#2)	0	Enabled
			1	Disabled
Cutting feed	Rapid traverse	SCR(No.11247#3)	0	Enabled
			1	Disabled
	Cutting feed	None	-	Enabled
	No movement	SCN(No.11247#4)	0	Disabled
			1	Enabled
No movement	Rapid traverse	SNR(No.11247#5)	0	Disabled
			1	Enabled
	Cutting feed	SNC(No.11247#6)	0	Enabled
			1	Disabled
	No movement	SNN(No.11247#7)	0	Enabled
			1	Disabled

- Example 1 of overlap [Rapid traverse ⇒ Cutting feed ⇒ Rapid traverse]

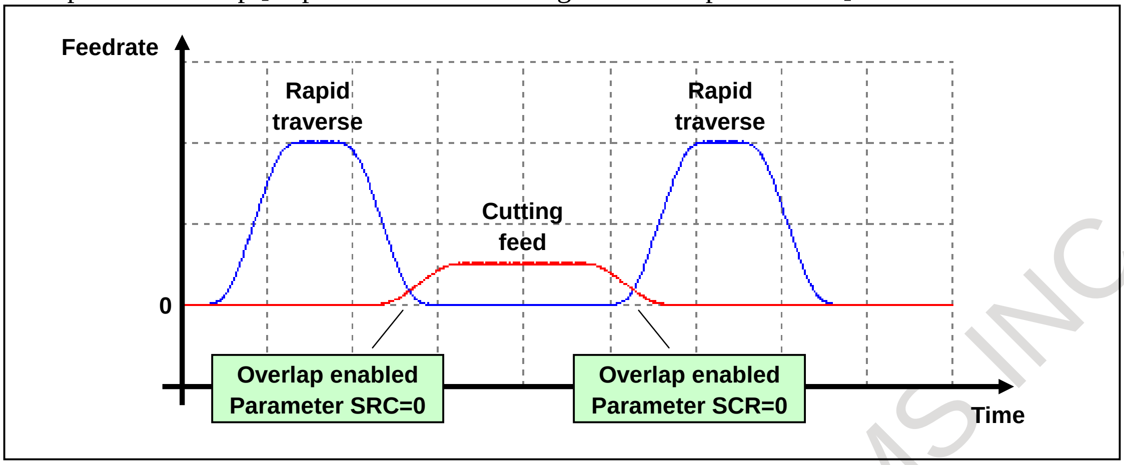


Fig. 7.2.1.4 (f) Example 1 of overlap

- Example 2 of overlap [Rapid traverse ⇒ Cutting feed ⇒ Rapid traverse]
(In case that overlap is disabled from rapid traverse block to cutting feed block)

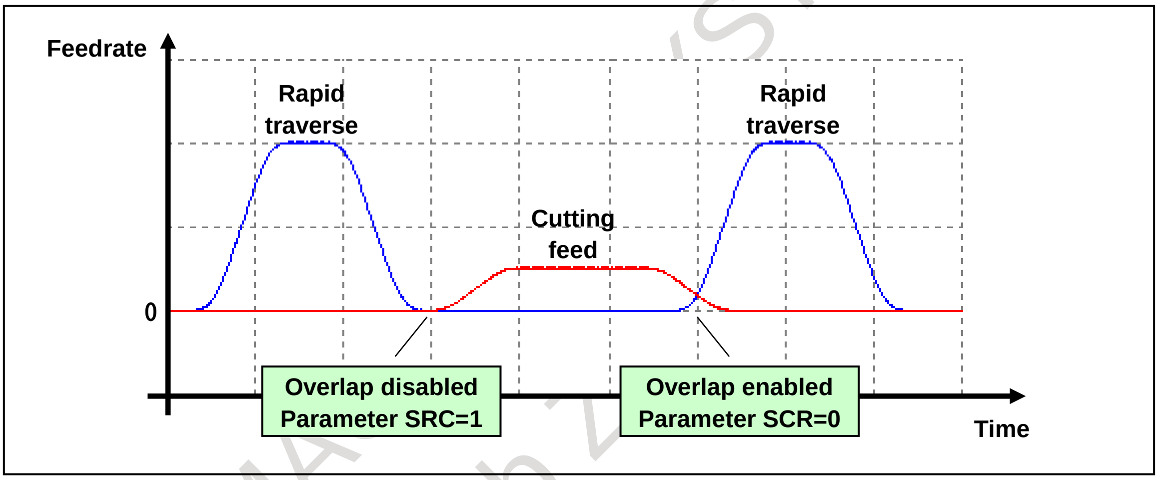


Fig. 7.2.1.4 (g) Example 2 of overlap

- Example 3 of overlap [Rapid traverse ⇒ No movement (only M code) ⇒ Cutting feed]
(In case that the execution time of M code is longer than overlap time)

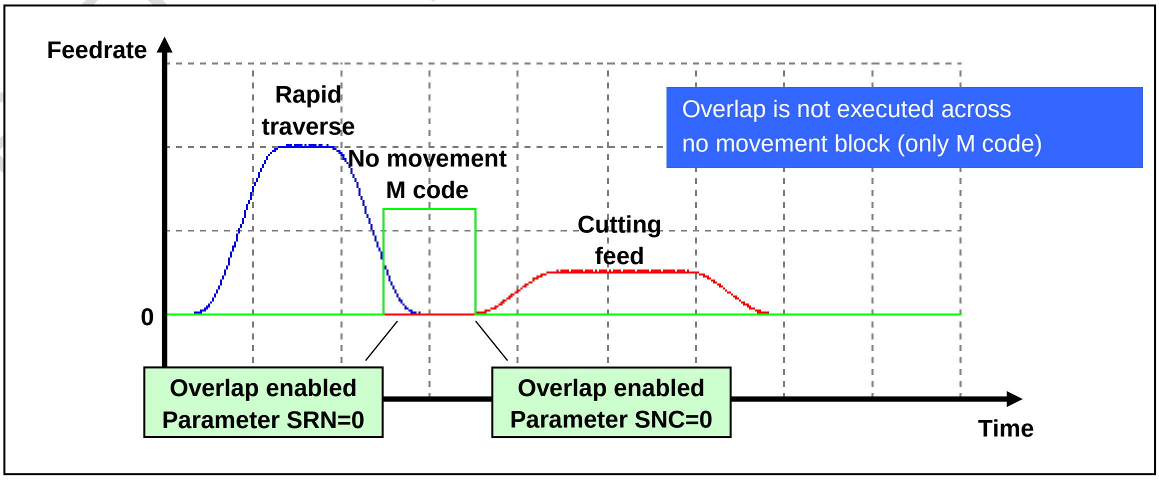


Fig. 7.2.1.4 (h) Example 3 of overlap

- Example 4 of overlap [Rapid traverse ⇒ No movement (only M code) ⇒ Cutting feed]
(In case that the execution time of M code is shorter than overlap time)

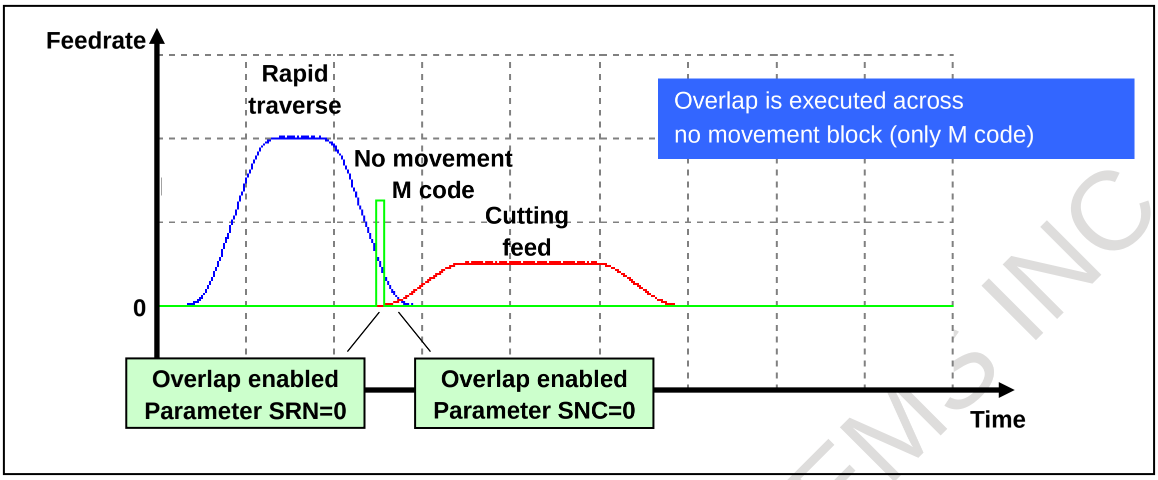


Fig. 7.2.1.4 (i) Example 4 of overlap

- Example 5 of overlap [Rapid traverse ⇒ No movement (only M code) ⇒ Cutting feed]
(In case that overlap is disabled from rapid traverse block to no movement block)

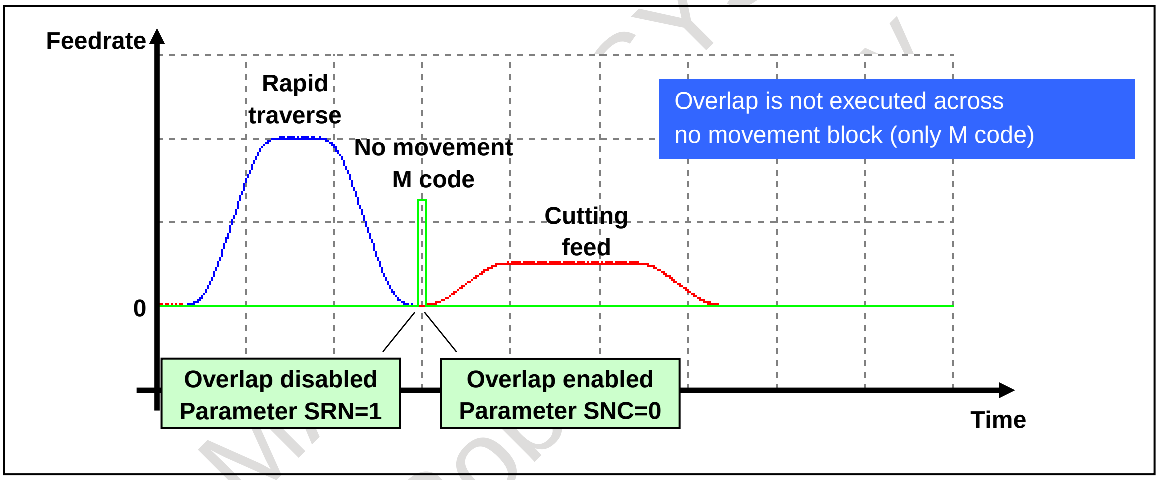


Fig. 7.2.1.4 (j) Example 5 of overlap

- Example 6 of overlap [Rapid traverse ⇒ No movement (only M code) ⇒ Cutting feed]
(In case that overlap is disabled from no movement block to cutting feed block)

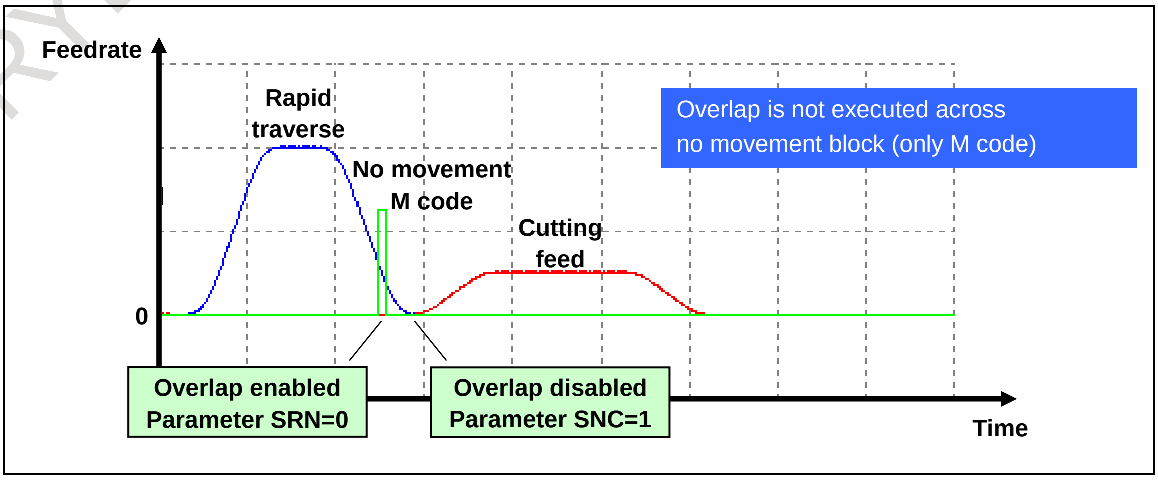


Fig. 7.2.1.4 (k) Example 6 of overlap

- Relation to each function

The relation to each function is as follows.

- Rapid traverse block overlap
  Rapid traverse block overlap can be used concurrently with Smart overlap. When rapid traverse block overlap and Smart overlap are used at the same time, overlap is performed from the bigger of the overlap value specified for the two functions.

- Optimum acceleration/deceleration for rigid tapping
  The function for acceleration/deceleration by Smart overlap is ineffective in Optimum acceleration/deceleration for rigid tapping block. Moreover, the overlap is not performed before and after the block.

- High precision oscillation function
  The function for acceleration/deceleration by Smart overlap is ineffective while High precision oscillation function is executing. Moreover, the overlap is not performed before and after the block.

- Axis synchronous control
  The function for acceleration/deceleration by Smart overlap is ineffective while Synchronization establishment function is executing. Moreover, the overlap is not performed before and after the block.

- PMC Axis Control
  Automatic acceleration/deceleration is ineffective while the skip command (03h) of PMC axis control is executing. The acceleration/deceleration by Smart overlap is ineffective too, and the overlap is not performed before and after the block.
  For the other commands of PMC axis control, the acceleration/deceleration by Smart overlap is effective, but the overlap is not performed before and after the block.

- Function of deceleration stop in case of power failure
  The function for acceleration/deceleration by Smart overlap is ineffective while the deceleration stop in case of power failure is executing. Moreover, the overlap is not performed before and after the block.

- Arbitrary speed threading
  The function for acceleration/deceleration by Smart overlap is ineffective while arbitrary speed threading is executing. Moreover, the overlap is not performed before and after the block.

- Non-buffering block
  The acceleration/deceleration by Smart overlap is effective in the non-buffering block. However, the overlap is ineffective before and after the block. In the following examples, the overlap is not performed between N1-N2 and between N2-N3.
  Example)
  N1 Gxx ;
  N2 Mxx ;···Non-buffering M code
  N3 Gxx ;

- Automatic reference position return (G28, G30)
  The acceleration/deceleration by Smart overlap is effective in the automatic reference position return command. However, the overlap is ineffective between the movement to the reference point and the next block. The overlap is effective between before the reference point return command block and the movement to the middle point.

- In-position check disable reference position return (G28.2,G30.2)
  This command is similar to automatic reference point return (G28,G30).

- High-speed G53 function (G53P1)
High-speed G53 function is non-buffering G code. But, the acceleration/deceleration by Smart overlap is effective, and the overlap is also effective before and after the block.

- Threading
The acceleration/deceleration by Smart overlap is effective in the start block of threading (Block that waits for one-rotation signal). However, the overlap is ineffective between previous block.

- Threading cycle
Smart overlap is performed between "retract" and "return" and between "return" and "positioning" in the threading cycle.

- Rigid tapping
The acceleration/deceleration by Smart overlap is effective in the following blocks. However, the overlap is not performed before and after the block.
  - Movement block from I point (Initial point) to R point and movement block from R point to Z point (hole bottom)
  - Movement block from R point to Z point (hole bottom) and movement block from Z point (hole bottom) to R point.
  - Movement block from Z point (hole bottom) to R point and movement block from R point to I point (Initial point)

- Exact Stop (G09)/Exact Stop mode (G61)
The function for acceleration/deceleration by Smart overlap is effective in the exact stop block. However, the overlap is not performed between the next block.

- Synchronous/Composite control, Superimposed control
If Smart overlap is effective even one axis, an operation equivalent to that of following setting is performed.
- Bit 5 (NCS) of parameter No.8160 is 1, bit 7 (NRS) of parameter No.8160 is 1, bit 1 (MSO) of parameter No.8168 is 1

- In-position check
If Smart overlap is effective even one axis, an operation equivalent to that of following setting is performed.
- Bit 4 of parameter No.11503 is 1

- In-position check signal
In Smart overlap between cutting feed blocks, the in-position check signal can be switched enabled and disabled by the parameter SIE(No.10735#4) .

- Manual operation
Smart overlap is also effective in manual operation.

- Dwell command
Between dwell command and the previous block, overlap can be switched between enabled and disabled by the parameter SOD(No.10735#7).

Disabling Smart overlap of rigid tap

When Smart overlap is applied to rigid tap, the sum of the time constant of acceleration/deceleration after interpolation(parameters No.5261 to 5264, etc) in rigid tap and the time constant of acceleration/deceleration after interpolation(parameters Nos.11248 and 11249) in Smart overlap is the total time. Therefore, it is necessary to subtract the time constant of acceleration/deceleration after interpolation in Smart overlap from the time constant of acceleration/deceleration after interpolation in rigid tap and adjust so that the total time constant of acceleration/deceleration after interpolation does not change. However, if the time constant of acceleration/deceleration after interpolation in rigid tap is less than the time constant of acceleration/deceleration after interpolation in Smart overlap, all the time constant of acceleration/deceleration after interpolation in Smart overlap cannot be subtracted from the time constant of acceleration/deceleration after interpolation in rigid tap. As a result, the total time constant of acceleration/deceleration after interpolation may be increased and the cycle time may be extended.

Table 7.2.1.4 (e) Adjustment example of time constant of acceleration/deceleration after interpolation

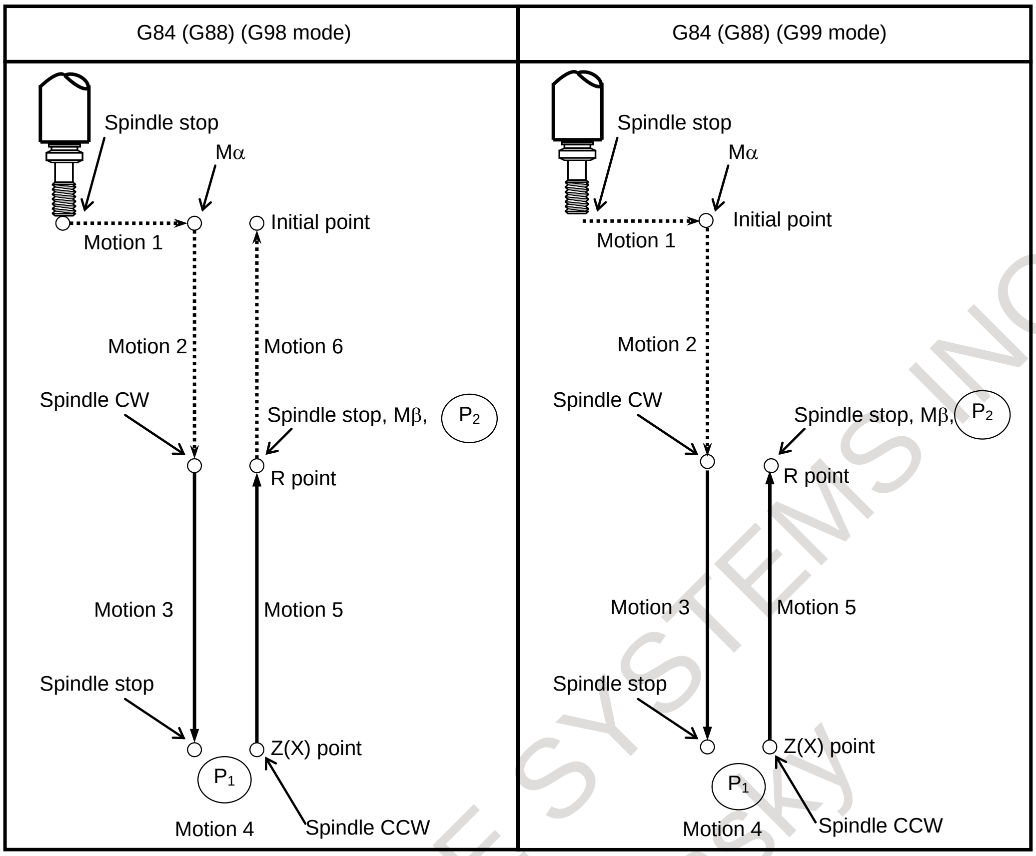
	Time constant of acceleration/deceleration after interpolation		Total time constant of acceleration/deceleration after interpolation
	rigid tap	Smart overlap	
Before Smart overlap applying	No.5261=16 No.5365=4	No.11248=0 No.11249=0	20
After Smart overlap application (Subtract can be done)	No.5261=0 No.5365=0	No.11248=16 No.11249=4	20
After Smart overlap application (Subtract cannot be done)	No.5261=0 No.5365=0	No.11248=24 No.11249=8	32

By setting bit 5 (SRI) or bit 6 (RIS) of parameter No. 10735 to 1, Smart overlap can be invalidated at the cutting and extraction in rigid tap. Therefore, the cycle time can be prevented being postponed in rigid tap.
For each rigid tap, the parameters of time constant of acceleration/deceleration after interpolation that is valid and the parameters to invalidate the Smart overlap are shown in the table below.

Table 7.2.1.4 (f) Parameters effective for each rigid tap

Type of rigid tap	Time constant of acceleration/deceleration after interpolation	Smart overlap invalid
Rigid tap	No.5261 to No.5264, No.5271 to No.5274, No.5365 to No.5368	Parameter SRI
Retraction for rigid tapping		
3-dimensional rigid tapping		
Retraction for 3-dimensional rigid tapping		
Rigid tapping by manual handle		
Rigid tapping of spindle control with servo motor	No.11060 to No.11063, No.11065 to No.11068, No.11052	Parameter RIS
Rigid tapping of spindle control with Cs contour control		
FSSB High-speed rigid tapping	Compliant with the base rigid tap	-

> **NOTE**
> - In synchronous control slave axis, setting of the path of the synchronous master axis is effective for the parameters SRI and RIS.
> - In composite control axes of composite control, setting of the path of the composite target axis is effective for the parameters SRI and RIS.



Rapid traverse
Cutting feed

P1 Dwell specified by address P in the program

P2 Dwell set by parameter No. 5111 (not performed in the FANUC Series 15 program format)

Mα Output of an M code for C-axis clamping

Mβ Output of an M code for C-axis unclamping (β=α+1) (not performed in the FANUC Series 15 program format)

Supplement: P1, Mα, Mβ, and P2, when not specified, are not executed and output.

When the parameter SIE or RIS is set to 0

In motion 3 and 5, the parameters (Nos.11248 and 11249) are added to the time constant of acceleration/deceleration after interpolation in rigid tap.

When the parameter SIE or RIS is set to 1

In motion 3 and 5, the parameters (Nos.11248 and 11249) are not added to the time constant of acceleration/deceleration after interpolation in rigid tap.

NOTE

Even when parameters SRI and RIS are set to 1, there is no change in motion 1, 2, 4, 6.

NOTE

NOTE

1. Smart overlap is effective for the rigid tapping and threading, etc. Therefore, please confirm the screw precision.
2. Smart overlap is effective for the skip command. Therefore, the skip coordinates value is influenced. So, when the accumulated pulses by acceleration/deceleration of Smart overlap is considered and compensated, set bit 1 (SEB) of parameter (No. 6201) to 1.
3. When the overlap is executed at the block where the direction of the movement reverses, the direction of the movement reverses before the tool reaches the programmed position. In the drilling, the tool doesn't reach the programmed bottom of a hole.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
10735	SOD	RIS	SRI	SIE				ARF

[Input type] Parameter input
[Data type] Bit path

#0 ARF In Smart overlap, between movement block of reference position return to the middle point and previous block overlap is :
0: Enabled.
1: Disabled.

#4 SIE In the axis that Smart overlap is effective, the in-position check signal is:
0: Disabled.
1: Enabled.

NOTE

Smart overlap is effective between the following blocks, but the in-position check signal is not effective.
- Rapid traverse - Rapid traverse
- Rapid traverse - Cutting feed
- Rapid traverse - No movement
- Cutting feed - Rapid traverse
- Cutting feed - No movement
- No movement - Rapid traverse
- No movement - Cutting feed
- No movement - No movement

#5 SRI In cutting and extraction in the rigid tap, Smart overlap is:
0: Depending on the setting in bit 1 (SOVx) of parameter No.11236
1: Disabled.

NOTE

This parameter is effective for the next rigid tap.

- Rigid tap
- Rigid tapping by manual handle
- 3-dimensional rigid tapping
- Retraction for rigid tapping
- Retraction for 3-dimensional rigid tapping

#6 RIS In cutting and extraction in the rigid tapping of spindle control with servo motor Smart overlap is:

0: Depend on the bit 1 (SOVx) of parameter No.11236 setting.
1: Disabled.

#7 SOD In Smart overlap, between dwell block and previous block overlap is :

0: Enabled.
1: Disabled.

10740	Maximum overlap value of Smart overlap for each axis（Rapid traverse - Rapid traverse)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

The maximum clamp value of Smart overlap between the rapid traverse block and the rapid traverse block is set.
The overlap value is clamped by the value set to this parameter. So, the inner deviation in the corner can be suppressed.

NOTE

When a value less than 0 are set, the specification of 0 is assumed.
In this case, the overlap by Smart overlap is not clamped.
When you clamp overlap value, set the value of in-position width or more.

10741	Maximum overlap value of Smart overlap for each axis（Rapid traverse – Cutting feed)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

The maximum clamp value of Smart overlap between the rapid traverse block and the cutting feed block is set.
The overlap value is clamped by the value set to this parameter. So, the inner deviation in the corner can be suppressed.

NOTE

When a value less than 0 are set, the specification of 0 is assumed.
In this case, the overlap by Smart overlap is not clamped.
When you clamp overlap value, set the value of in-position width or more.

10742	Maximum overlap value of Smart overlap for each axis (Cutting feed - Rapid traverse)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

The maximum clamp value of Smart overlap between the cutting feed block and the rapid traverse block is set.
The overlap value is clamped by the value set to this parameter. So, the inner deviation in the corner can be suppressed.

NOTE
When a value less than 0 are set, the specification of 0 is assumed.
In this case, the overlap by Smart overlap is not clamped.
When you clamp overlap value, set the value of in-position width or more.

	#7	#6	#5	#4	#3	#2	#1	#0
11236							SOVx	

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#1 SOVx Smart overlap is
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
11247	SNN	SNC	SNR	SCN	SCR	SRN	SRC	SRR

[Input type] Parameter input
[Data type] Bit path

#0 SRR In Smart overlap, between rapid traverse block and rapid traverse block overlap is :
0: Enabled.
1: Disabled.

#1 SRC In Smart overlap, between rapid traverse block and cutting feed block overlap is :
0: Enabled.
1: Disabled.

#2 SRN In Smart overlap, between rapid traverse block and non-movement block overlap is :
0: Enabled.
1: Disabled.

#3 SCR In Smart overlap, between cutting feed block and rapid traverse block overlap is :
0: Enabled.
1: Disabled.

#4 SCN In Smart overlap, between cutting feed block and non-movement block overlap is :
0: Enabled.
1: Disabled.

#5 SNR In Smart overlap, between non-movement block and rapid traverse block overlap is :
0: Enabled.
1: Disabled.

#6 SNC In Smart overlap, between non-movement block and cutting feed block overlap is :
0: Enabled.
1: Disabled.

#7 SNN In Smart overlap, between non-movement block and non-movement overlap is :
0: Enabled.
1: Disabled.

11248	Time constant 1 of acceleration/deceleration after interpolation in Smart overlap for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 64

Set the time constant 1 of acceleration/deceleration after interpolation in Smart overlap for each axis.

11249	Time constant 2 of acceleration/deceleration after interpolation in Smart overlap for each axis

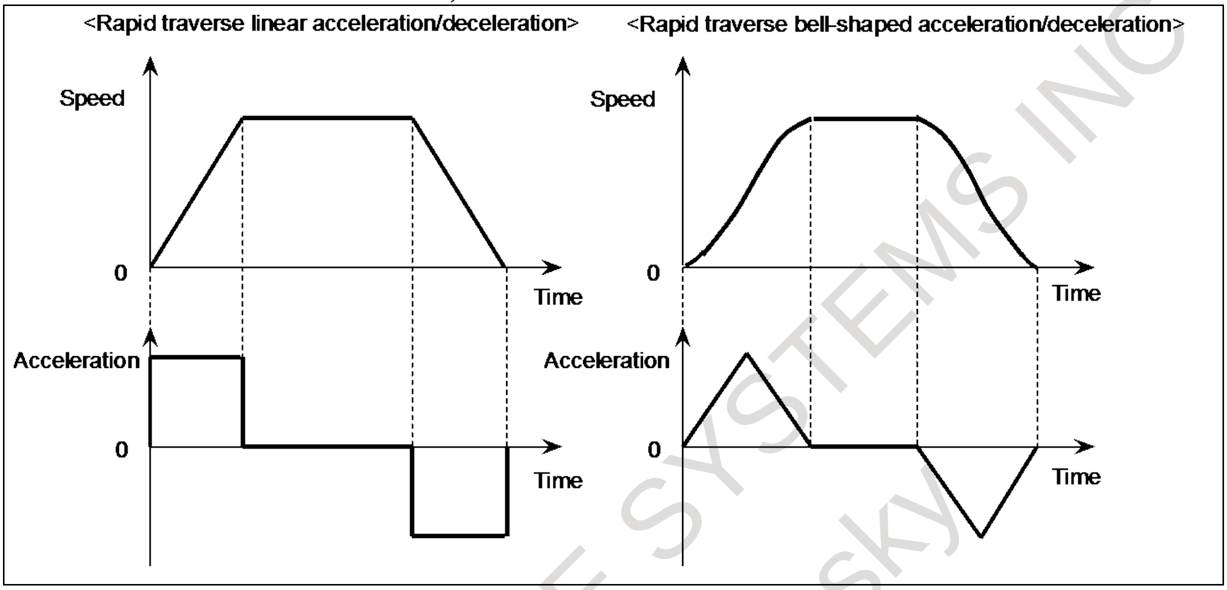
[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 64

Set the time constant 2 of acceleration/deceleration after interpolation in Smart overlap for each axis for each axis.

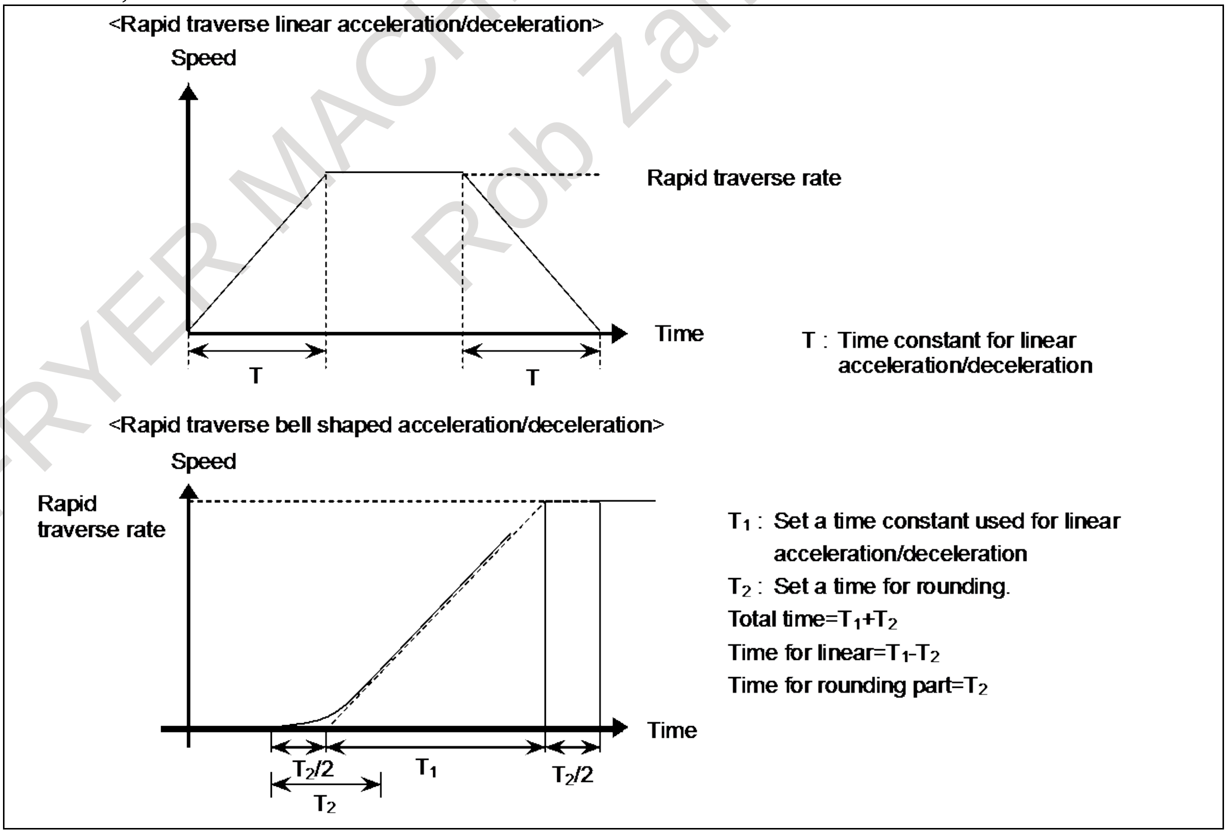
7.2.2 Rapid Traverse Bell-shaped Acceleration/Deceleration

Overview

Rapid traverse bell-shaped acceleration/deceleration smoothly increases or decreases the rapid traverse rate, reducing the stress and strain imposed on the machine due to the variation in the acceleration with changes in the feedrate. As the time constant for bell-shaped acceleration/deceleration can be smaller than that for linear acceleration/deceleration, the time needed for acceleration/ deceleration can be reduced.



This function is enabled when the time constants for rapid traverse bell-shaped acceleration/deceleration T_1 and T_2 are specified in parameters Nos. 1620 and 1621, respectively.
In this case, the time needed for acceleration/deceleration is as follows.



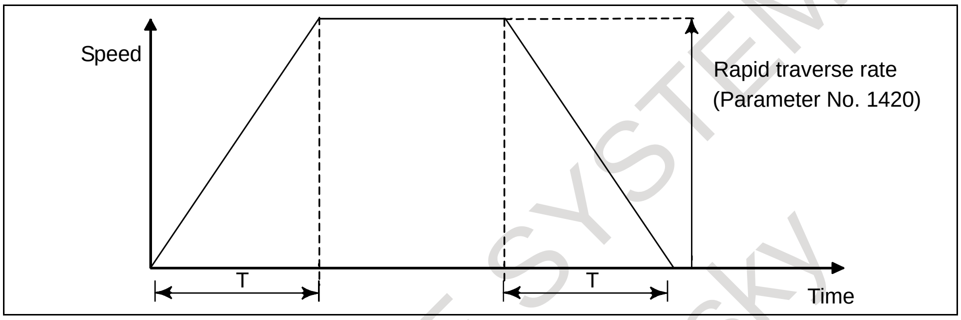
Set a time when rapid traverse override is 100%. When it is less than 100%, the total time is reduced (constant acceleration method).
Value of T_1 is determined from motor torque. Set a value of T_2 to 24 msec or 32 msec.

Parameter

1620	Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis

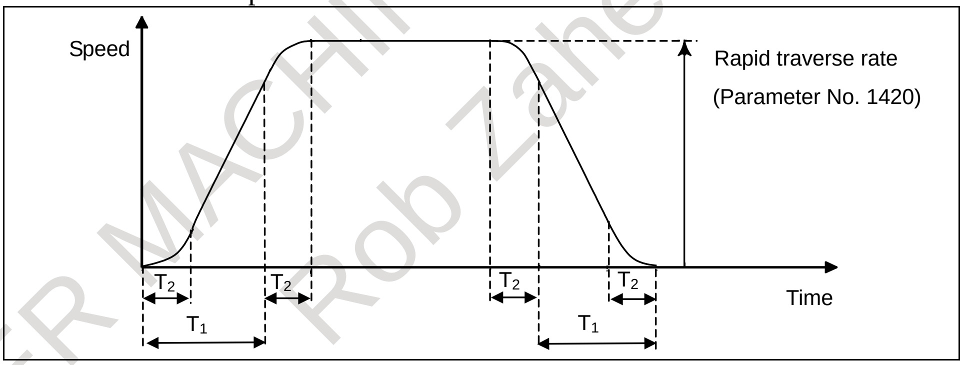
[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Specify a time constant used for acceleration/deceleration in rapid traverse.
[Example]
For linear acceleration/deceleration



T : Setting of parameter No. 1620

For bell-shaped acceleration/deceleration



T_1 : Setting of parameter No. 1620
T_2 : Setting of parameter No. 1621
(However, $T_1 \geq T_2$ must be satisfied.)
Total acceleration (deceleration) time : $T_1 + T_2$
Time for linear portion : $T_1 - T_2$
Time for curve portion : $T_2 \times 2$

1621	Time constant T_2 used for bell-shaped acceleration/deceleration in rapid traverse for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 512

Specify time constant T_2 used for bell-shaped acceleration/ deceleration in rapid traverse for each axis.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	Automatic acceleration/deceleration

7.2.3 Linear Acceleration/Deceleration after Cutting Feed Interpolation

Overview

If linear Acceleration/Deceleration after Cutting Feed Interpolation is enabled (if bit 0 (CTL) of parameter No. 1610 is set to 1), acceleration/ deceleration is performed as follows:

Cutting feed:

Linear acceleration/deceleration (constant acceleration time)

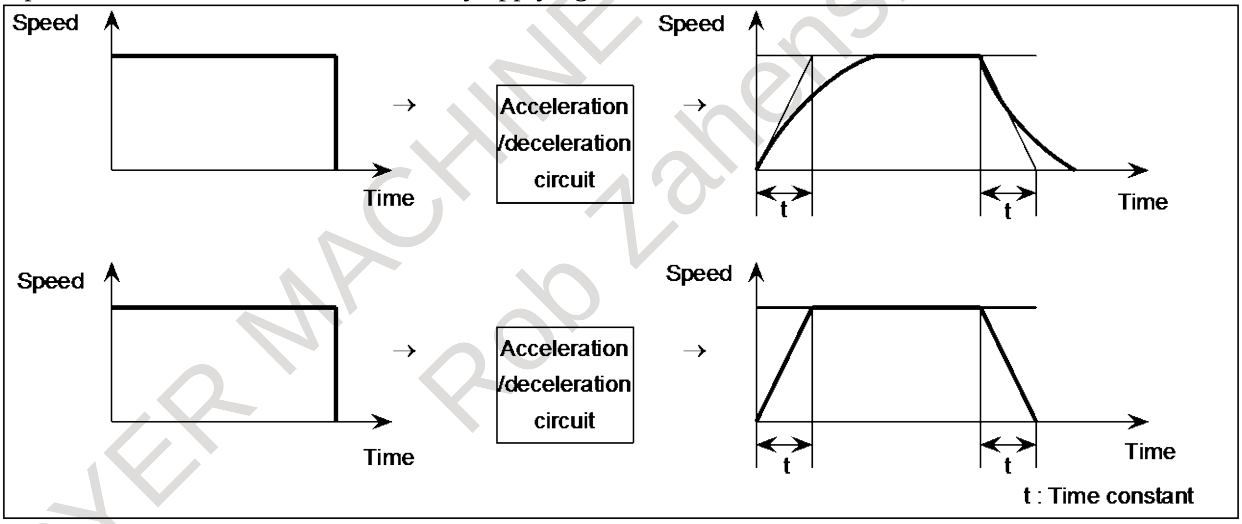
Specify the acceleration/deceleration time constant for each axis in parameter No. 1622.

Jog feed:

Exponential or linear acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1624.

If the time constant for linear acceleration/deceleration is the same as that for exponential acceleration/deceleration, the delay of linear acceleration/deceleration in relation to a command is 1/2 of the delay of exponential acceleration/deceleration, so that the time required for acceleration and deceleration can be reduced. If circular interpolation is performed, especially when high-speed cutting is being performed, the actual tool path created after acceleration/deceleration will deviate from the programmed arc in the radial direction. This deviation can also be reduced, in comparison with exponential acceleration/deceleration, by applying linear acceleration/deceleration.



This function is enabled when bit 0 (CTL) of parameter No. 1610 is set to 1. If bell-shaped Acceleration/Deceleration after Cutting Feed Interpolation is also enabled, bell-shaped acceleration/deceleration is executed. The time constants for cutting feed and jog feed for each axis are specified in parameters Nos. 1622 and 1624 respectively, in the same way as for exponential acceleration/deceleration. The values specified for the FL feedrate for cutting feed (parameter No. 1623) and the FL feedrate for jog feed (parameter No. 1625) are ignored (always assumed to be 0).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx				CTLx

[Input type] Parameter input
[Data type] Bit axis

#0 CTLx Acceleration/deceleration in cutting feed or dry run during cutting feed
0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied.

> **NOTE**
> This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

#4 JGLx Acceleration/deceleration in jog feed
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied.
(Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)

> **NOTE**
> This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

1622	Time constant of acceleration/deceleration in cutting feed for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in cutting feed, bell-shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation in cutting feed for each axis. Which type to use is selected with bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610. Except for special applications, the same time constant must be set for all axes in this parameter. If the time constants set for the axes differ from each other, proper straight lines and arcs cannot be obtained.

1624	Time constant of acceleration/deceleration in jog feed for each axis.

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

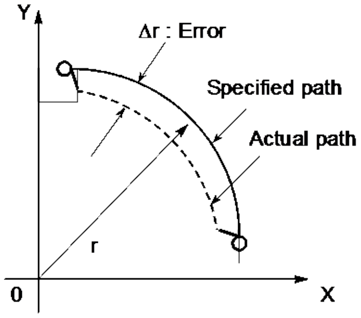
Set the time constant used for acceleration/deceleration in jog feed for each axis.

Note

NOTE

1 If linear Acceleration/Deceleration after Cutting Feed Interpolation is enabled, linear acceleration/deceleration is executed during cutting feed and during a dry run. Linear acceleration/deceleration can also be executed during jog feed if the bit 4 (JGL) of parameter No. 1610 is specified accordingly.

2 In circular interpolation especially when circular cutting is executed at high speed, the actual path of the accelerated or decelerated tool deviates from the specified arc in the direction of the radius.



Δr : Maximum radius error (mm)
v : Feedrate (mm/s)
r : Radius of arc (mm)
T_1 : Acceleration/deceleration after interpolation time constant cutting (sec)
T_2 : Time constant of servo motor (sec) (Inverse of position loop gain)

The maximum error in the radial direction (Δr) can be approximated by the following expressions:

$$\Delta r = \left(\frac{1}{2}T_1^2 + \frac{1}{2}T_2^2\right)\frac{V^2}{r}$$ …Exponential acceleration/deceleration

$$\Delta r = \left(\frac{1}{24}T_1^2 + \frac{1}{2}T_2^2\right)\frac{V^2}{r}$$ …Linear acceleration/deceleration after interpolation

$$\Delta r = \left(\frac{1}{48}T_1^2 + \frac{1}{2}T_2^2\right)\frac{V^2}{r}$$ …Bell-shaped acceleration /deceleration after interpolation

If the error caused by the time constant of the servo motor is excluded, the error cause by linear acceleration/deceleration after interpolation is 1/12 of that caused by exponential acceleration/deceleration. The radius error in the case of bell-shaped acceleration/deceleration after interpolation is a half of linear acceleration/deceleration after interpolation.

3 Linear acceleration/deceleration can be executed both for cutting feed and for jog feed along a PMC axis. Acceleration/deceleration for cutting feed is executed even if acceleration/deceleration for jog feed is selected. In jog feed along the PMC axis, the time constant for cutting feed is used instead of that for jog feed.

7.2.4 Bell-Shaped Acceleration/Deceleration after Cutting Feed Interpolation

Overview

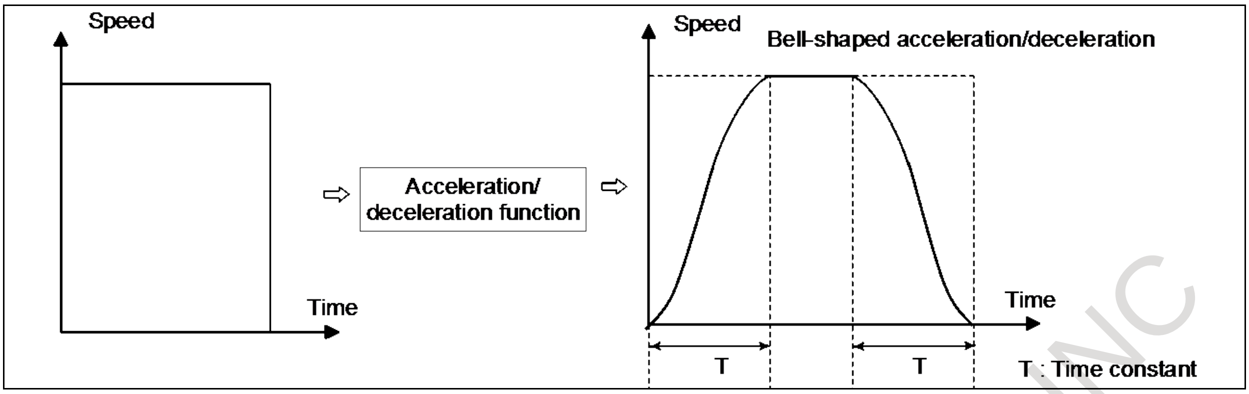
The bell-shaped acceleration/deceleration after cutting feed interpolation provides smooth acceleration and deceleration to reduce stress and strain on the machine. If this function is enabled (if bit 1 (CTB) of parameter No. 1610 is set to 1), acceleration/deceleration is performed as follows:

Cutting feed:

Bell-shaped acceleration/deceleration (constant acceleration time)
Specify the acceleration/deceleration time constant for each axis in parameter No. 1622.

Jog feed:

Exponential or bell-shaped acceleration/deceleration (constant acceleration time)
Specify the acceleration/deceleration time constant for each axis in parameter No. 1624.



Bell-shaped acceleration/deceleration after cutting feed interpolation is enabled when bit 1 (CTB) of parameter No.1610 is specified. The time constants for cutting feed and for jog feed for each axis are specified in parameters Nos. 1622 and 1624 respectively, in the same way as exponential acceleration/deceleration. The values specified for the FL feedrate for cutting feed (parameter No. 1623) and the FL feedrate for jog feed (parameter No. 1625) are ignored (always assumed to be 0).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx			CTBx	CTLx

[Input type] Parameter input
[Data type] Bit axis

#0 CTLx Acceleration/deceleration in cutting feed or dry run during cutting feed
0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied.

> **NOTE**
> This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

#1 CTBx Acceleration/deceleration in cutting feed or dry run during cutting feed
0: Exponential acceleration/deceleration or linear acceleration/ deceleration is applied. (depending on the setting in bit 0 (CTLx) of parameter No. 1610)
1: Bell-shaped acceleration/deceleration is applied.

> **NOTE**
> This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

#4 JGLx Acceleration/deceleration in jog feed
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied. (Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)

> **NOTE**
> This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

1622	Time constant of acceleration/deceleration in cutting feed for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in cutting feed, bell-shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation in cutting feed for each axis. Which type to use is selected with bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610. Except for special applications, the same time constant must be set for all axes in this parameter. If the time constants set for the axes differ from each other, proper straight lines and arcs cannot be obtained.

1624	Time constant of acceleration/deceleration in jog feed for each axis.

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set the time constant used for acceleration/deceleration in jog feed for each axis.

Note

> **NOTE**
> 1 If bell-shaped acceleration/deceleration after interpolation during cutting feed is enabled, bell-shaped acceleration/deceleration is executed during cutting feed and during a dry run. Bell-shaped acceleration/deceleration can also be executed during jog feed if the bit 4 (JGL) of parameter No. 1610 is specified accordingly.
> 2 In circular interpolation, the actual tool path after acceleration/deceleration deviates from the programmed arc in the radial direction. To overcome this radial deviation, see the note on linear Acceleration/Deceleration after Cutting Feed Interpolation.
> 3 Bell-shaped acceleration/deceleration can be executed both for cutting feed and for jog feed along a PMC axis. The time constant for acceleration/deceleration for jog feed is the same as that for cutting feed. In jog feed along the PMC axis, the time constant for cutting feed is used instead of that for jog feed.

7.2.5 Optimum Torque Acceleration/Deceleration

Overview

This function enables acceleration/deceleration in accordance with the torque characteristics of the motor and the characteristics of the machines due to its friction and gravity.
Usually, because of the friction of the machine, gravity, the torque characteristics of the motor, and other factors, the acceleration/ deceleration performance (torque for acceleration/deceleration) is different with direction of movement, acceleration or deceleration. In this function, acceleration pattern of rapid traverse for the following situations, plus movement and acceleration, plus movement and deceleration, minus movement and acceleration, minus movement and deceleration can be set into parameters according to the torque for acceleration/deceleration of each situation. Acceleration/deceleration can be performed according to these parameter setting, so that the most of the capability of the motor can be used and positioning time can be reduced.

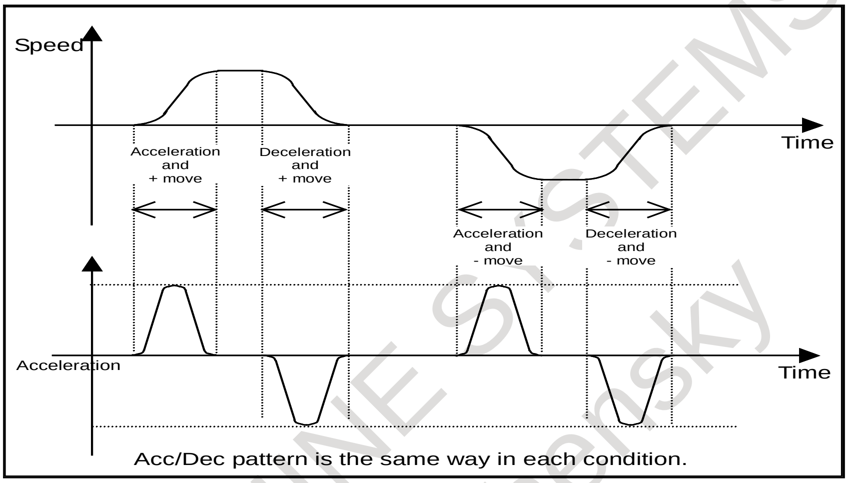


Fig. 7.2.5 (a) Conventional acceleration/deceleration

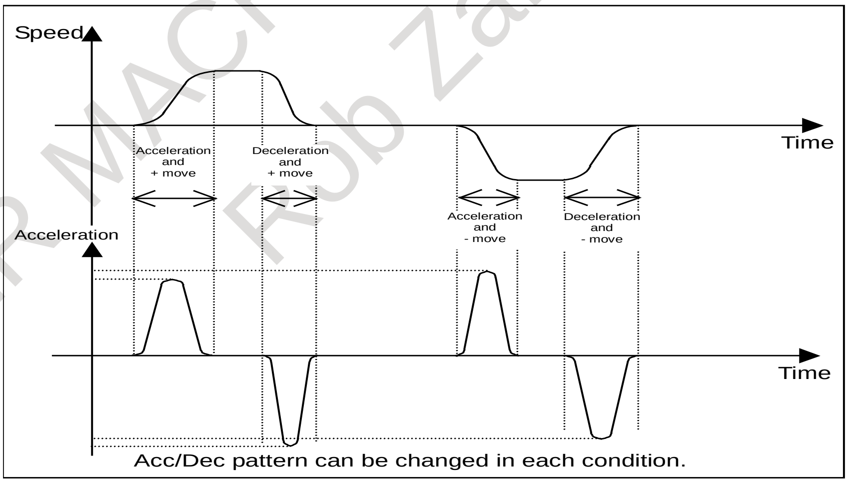


Fig. 7.2.5 (b) Acceleration/deceleration with this function

Explanation

Optimum torque acceleration/deceleration selects the acceleration pattern set with parameters on the basis of the axial movement direction and the acceleration/deceleration state, determines the acceleration for each axis from the current speed, and controls the tangential acceleration/deceleration for rapid traverse.

- Setting optimum torque acceleration/deceleration

When bit 0 (FAP) of parameter No. 19540 and bit 1 (LRP) of parameter No. 1401 are set to 1, and a value other than 0 is set for any one of the axes as the reference acceleration in parameter No. 1671 as shown in the Fig. 7.2.5 (a), rapid traverse is accelerated/decelerated by optimum torque acceleration/deceleration.

Table 7.2.5 (a) Optimum torque acceleration/deceleration

Bit 0 (FAP) of parameter No. 19540 (Optimum torque acc/dec)	Bit 1 (LRP) of parameter No. 1401 (Linear type positioning)	Reference acceleration	Bell-shaped acceleration change time	Acceleration pattern
1	1	No.1671	No.1672	See "Setting an acceleration pattern".

To enable bell-shaped acceleration/deceleration in addition to optimum torque acceleration/deceleration, set the bell-shaped acceleration change time with parameter No. 1672.

NOTE
To use optimum torque acceleration/deceleration, set bit 5 (FRP) of parameter No. 19501 to 1.

- Required conditions in addition to parameter setting

If the conditions for performing AI contour control and for look-ahead acceleration/deceleration before interpolation are satisfied, optimum torque acceleration/deceleration is performed for accelerating and decelerating rapid traverse. However, when bit 0 (FAE) of parameter No. 11240 is set to 1, optimum torque acceleration/deceleration is enabled even if AI contour control is disabled. If rapid traverse is subject to optimum torque acceleration/deceleration, after-interpolation acceleration/deceleration does not apply to rapid traverse.

- Cases in which optimum torque acceleration/ deceleration is disabled

If optimum torque acceleration/deceleration is disabled by parameter settings, or the required conditions other than parameter settings are not satisfied, acceleration/deceleration after interpolation is used to accelerate and decelerate rapid traverse.

- Setting acceleration pattern data

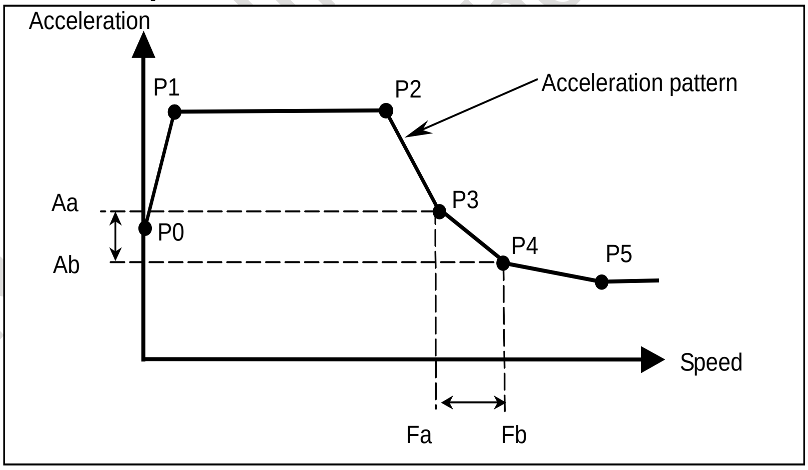


Fig. 7.2.5 (c) Setting acceleration pattern

Set the speed and the acceleration at each of the acceleration setting points P0 to P5 for each condition, plus movement and acceleration, plus movement and deceleration, minus movement and acceleration, minus movement and deceleration, and for each axis.
The line joining the acceleration setting points is regarded a acceleration pattern.
For example, while the speed is between Fa and Fb in the Fig. 7.2.5 (c), the acceleration is calculated with Aa and Ab. Tangential acceleration is controlled not to exceed the calculated acceleration for each axis.

> **NOTE**
> It is not desirable to set an acceleration pattern in which a large acceleration is set right at a speed of 0, because this will shock the machine. So, **be sure to set an acceleration pattern so that a relatively low acceleration rate is applied at speed 0** as shown Fig. 7.2.5 (c).

Table 7.2.5 (b) Parameters for acceleration pattern

Acceleration setting point	Speed parameter	Acceleration parameter			
		During acceleration		During deceleration	
		During movement in plus direction	During movement in minus direction	During movement in plus direction	During movement in minus direction
P0	(Speed 0)	No.19545	No.19551	No.19557	No.19563
P1	No.19541	No.19546	No.19552	No.19558	No.19564
P2	No.19542	No.19547	No.19553	No.19559	No.19565
P3	No.19543	No.19548	No.19554	No.19560	No.19566
P4	No.19544	No.19549	No.19555	No.19561	No.19567
P5	No.1420	No.19550	No.19556	No.19562	No.19568

The speed at P0 is 0, and the speed at P5 is the rapid traverse rate specified with parameter No. 1420. The speeds at P1 to P4 are to be set into speed parameters Nos. 19541 to 19544 as ratio to the rapid traverse speed (parameter No. 1420).
Any acceleration setting point for which the speed parameter (one of Nos. 19541 to 19544) is set to 0 will be skipped, and the next point whose speed parameter is set to a non-zero value will be joined together connected with line. The polygonal line is regarded as acceleration pattern.
The accelerations at P0 to P5 are to be set into acceleration parameters Nos. 19545 to 19568 as ratio to the reference acceleration. If any of the acceleration parameters Nos. 19545 to 19568 is set to 0, the acceleration is assumed 100% (Reference acceleration). Acceleration parameters should be set to 0 at the acceleration setting point whose speed parameter is set to 0.
If this function is enabled and parameter No. 1671 for an axis are set to 0, the following values are assumed as the reference acceleration for that axis:
1000.0mm/sec^2, 100.0inch/sec^2, 100.0 degree/sec^2

- Example of setting acceleration pattern data

In this example, the machine is equipped with the α*i*S 30/4000.
Motor speed at rapid traverse is 3000 (min^{-1}).

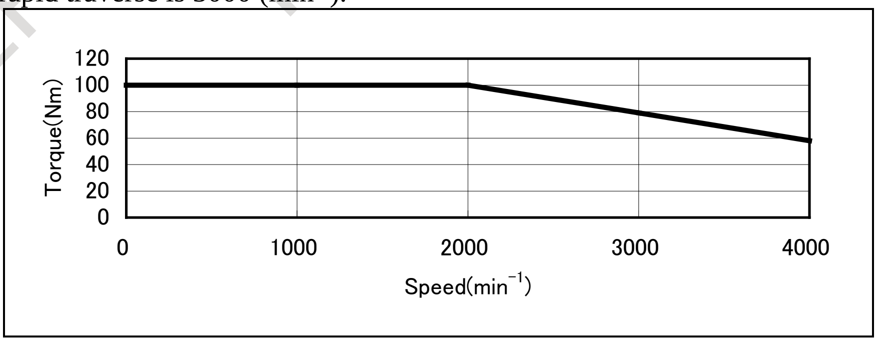


Fig. 7.2.5 (d) Speed-torque characteristics of model α*i*S 30/4000

Specifications of the motor model α*i*S 30/4000

Rotor inertia	:0.0099(Kgm^2)	
Maximum torque	:100(Nm)	Speed 0 to 2000(min^{-1})
Torque at rapid traverse	:79(Nm)	Speed 3000(min^{-1})
Minimum torque	:58(Nm)	Speed 4000(min^{-1})

It assumes that 10 (Nm) is needed for the friction torque, so that the torque for acceleration/deceleration is shown as the following figure.
Because the friction torque is different on each machine, it is necessary to observe the actual torque output on the machine for deciding acceleration pattern.

Maximum torque	:90(=100-10)(Nm)	Speed 0 to 2000(min^{-1})
Torque at rapid traverse	:69(=79-10)(Nm)	Speed 3000(min^{-1})
Minimum torque	:48(=58-10)(Nm)	Speed 4000(min^{-1})

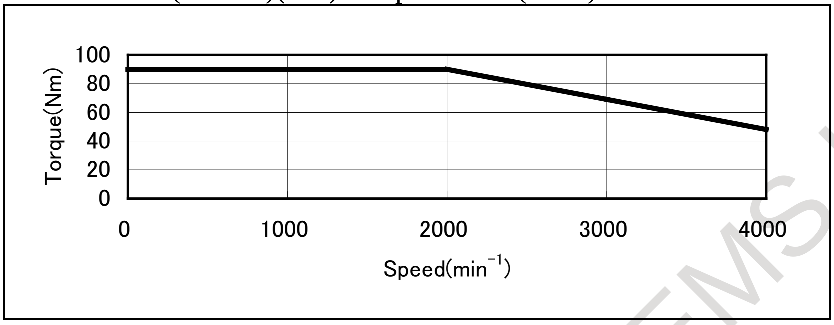


Fig. 7.2.5 (e) Torque for Acc/Dec with consideration of friction

Let the torque be x (Nm), the inertia be y(Kgm^2), and the ball screw pitch p(mm), then the acceleration A is calculated as follows:

$$A=\frac{x[N\cdot m]}{y[kg\cdot m^2]}\times\frac{p}{2\pi}[mm]=\frac{x([kg\cdot m/\sec^2][m])}{y[kg\cdot m^2]}\times\frac{p}{2\pi}[mm]$$

$$=\frac{x\times p}{2\pi\times y}[mm/\sec^2]$$

Machine specification is assumed as follows,

Ball screw pitch : 16 (mm)
Inertia : The machine inertia is to be 2.0 times higher than that of the rotor.
(Rotor inertia: 0.0099 (Kgm^2))

Assumption magnification as inertia of rotor and machine : 3.0 times
According to the above equation of torque and acceleration, the acceleration at maximum torque is,

$$\frac{90\times 16}{2\pi\times 3.0\times 0.0099}=7717[mm/\sec^2]$$

The acceleration when torque 69 (Nm) at rapid traverse rate (3000 min^{-1}) is,

$$\frac{69\times 16}{2\pi\times 3.0\times 0.0099}=5916[mm/\sec^2]$$

From the above data, the parameters related to acceleration pattern is given in the Table 7.2.5 (c).
The example assumes that the torque for acceleration/deceleration is the same regardless of whether acceleration or deceleration is in progress or whether the movement is in the plus or minus direction.

- When setting the speed at P1, use the following value as a rough guide:
  If a bell-shaped acceleration change time (parameter No. 1672) is set, use as a rough guide the ratio of the following speed to the rapid traverse rate:
  Reference acceleration × T2/4
  where T2 is the bell-shaped acceleration change time (msec).
  For example, assume T2 to be 40 msec, then the speed at P1 should be approximately
  4124 [mm/sec^2]×40 [msec]/4
  =4124 [mm/sec^2]×602×40 [msec]/60000/4
  =2474 [mm/min]
- If no bell-shaped acceleration change time (parameter No. 1672) is set, use about 5% of the rapid traverse rate as a rough guide.

Table 7.2.5 (c) Example of setting parameters related to acceleration pattern

	Parameter number	Setting	Unit	Remarks
Rapid traverse rate	1420	48000.	mm/min	The ball screw pitch is assumed 16 mm, so that the rapid traverse rate is 48000 mm/min at the maximum speed 3000 (min^{-1}).
Reference acceleration	1671	4124.	msec	Reference acceleration is 4124mm/sec^2.
Speed at P1	19541	515	0.01%	- Assuming that parameter No. 1672 (bell-shaped acceleration change time) is set to 40 (msec), the speed at P1 will be 2474 mm/min from the calculation described on the previous page. Set its ratio to the rapid traverse rate, or 5.15%. 0.0515=2474/48000 - If parameter No. 1672 (bell-shaped acceleration change time) is not set, set about 5.00%.
Speed at P2	19542	6666	0.01%	Since the torque characteristic is constant (90 Nm) in speeds reaching 2000 min^{-1}, set P2 to the ratio of the speed (32000 mm/min) at 2000 min^{-1} to the rapid traverse rate (48000 mm/min). 0.6666=32000 / 48000
Speeds at P3 to P4	19543 to 19544	0	0.01%	P3 to P4 are skipped because the torque drops almost linearly from the speed 2000 (min^{-1}) to 3000 (min^{-1}).
Acceleration at P0	19545,19551, 19557,19563	9356	0.01%	Set half the acceleration at P1, or 9356.
Acceleration at P1	19546,19552, 19558,19564	18712	0.01%	At P1, 90(Nm) can be used for the acceleration/deceleration, so set the ratio of the acceleration at 90 (Nm) to the reference acceleration. (1.8712 = 7717/4124)
Acceleration at P2	19547,19553 19559,19565	18712	0.01%	At P2, set the same speed as that at P1.
Acceleration at P3 to P4	19548 to 19549, 19554 to 19555, 19560 to 19561 19566 to 19567	0	0.01%	0 is set because P3 to P4 are skipped.
Acceleration at P5	19550,19556 19562,19568	14345	0.01%	At P5, 69(Nm) can be used for the acceleration/deceleration, so set the ratio of the acceleration at 69 (Nm) to the reference acceleration. (1.4345 = 5916 / 4124)

With the above parameter settings, the acceleration pattern will be shown as the Fig. 7.2.5 (f). From speeds from 0 mm/min to 2474 mm/min, the acceleration as calculated in accordance with the acceleration pattern is applied; from speeds from 2474 mm/min to 32000 mm/min, an acceleration of 7716 mm/sec^2; and from speeds from 32000 mm/min to 48000 mm/min, the acceleration as calculated in accordance with the acceleration pattern.

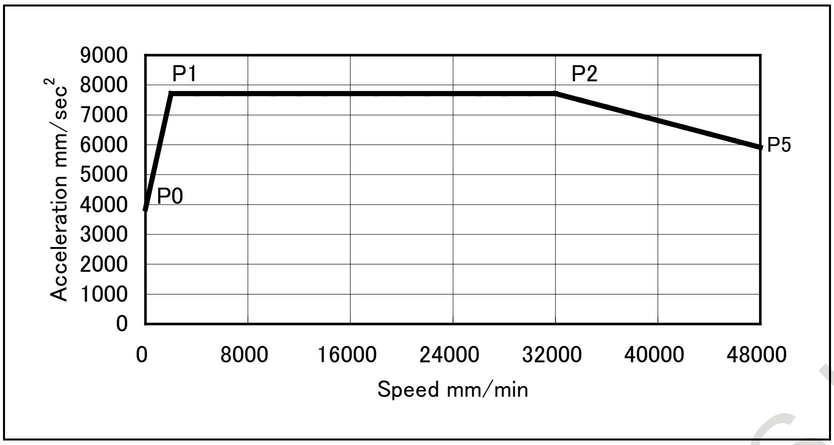


Fig. 7.2.5 (f) Acceleration pattern with consideration of friction

> **NOTE**
> The values in the model αiS 30/4000 speed-torque characteristic diagram are just typical ones. The values will change depending on the digital servo software, parameters, input voltage, and other factors.
> The optimum acceleration will, therefore, change due to the characteristics of the machine.

- Examples of setting if the torque of acceleration/deceleration differs depending on whether acceleration or deceleration is in progress and whether the movement is in the plus or minus direction

From the effect of gravity and friction, torque for acceleration/deceleration is different on each condition, such as acceleration, deceleration or plus move (up), minus move (down).
The following example is for the vertical axis and gravity and friction torque are assumed as follows.
Torque of gravity : 20 (Nm)
Torque of friction : 10 (Nm)
Because these value are different on each machine, it is necessary to observe the output torque on the actual machine for deciding acceleration pattern.
The conditions are the same as the previous example.
(Condition)

Motor speed at rapid traverse	: 3000(min^{-1})
Ball screw pitch	: 16(mm)
Inertia	: The machine inertia is to be 2.0 times higher than that of the rotor.
Rotor inertia	: 0.0099(Kgm^2)
Maximum motor torque	: 100(Nm) Speed 0 to 2000(min^{-1})
Motor torque at rapid traverse	: 79(Nm) Speed 3000(min^{-1})
Minimum motor torque	: 58(Nm) Speed 4000(min^{-1})

(1) In case of plus move (up) and acceleration
Because torque of Gravity and friction work against the output torque of motor, the torque for acceleration/deceleration is as follows.

Maximum torque	: 70(=100-20-10) (Nm)	Speed 0 to 2000(min^{-1})
Torque at rapid traverse	: 49(=79-20-10) (Nm)	Speed 3000(min^{-1})
Minimum torque	: 28(=58-20-10) (Nm)	Speed 4000(min^{-1})

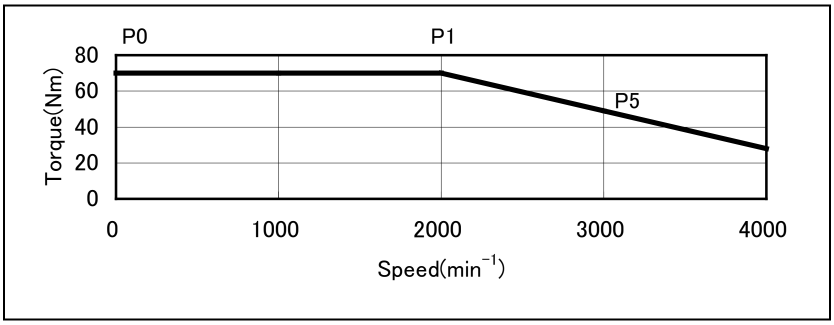


Fig. 7.2.5 (g) Torque for Acc/Dec in case of + move and acceleration with consideration of gravity and friction.

Parameter setting is as follows,

Fig.7.2.5 (d)

	Parameter number	Setting	Unit	Remarks
Acceleration at P0	19545	7277	0.01%	Set half the acceleration at P1, or 7277.
Acceleration at P1-P2	19546,19547	14554	0.01%	At P1 and P2, 70(Nm) can be used for the acceleration/ deceleration, so set the ratio of the acceleration at 70 (Nm) to the reference acceleration. (1.4554 = 6002 / 4124)
Acceleration at P3-P4	19548 to 19549	0	0.01%	0 is set because P3 to P4 are skipped.
Acceleration at P5	19550	10187	0.01%	At P5, 49(Nm) can be used for the acceleration/deceleration, so set the ratio of the acceleration at 49 (Nm) to the reference acceleration. (1.0187 = 4201 / 4124)

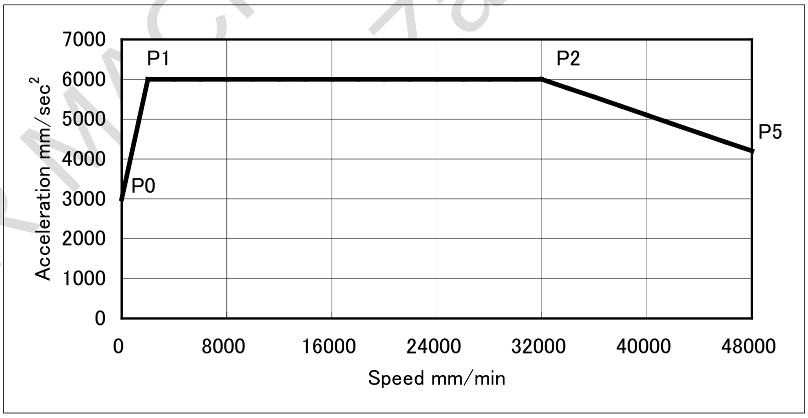


Fig. 7.2.5 (h) Acceleration pattern in case of + move and acceleration with consideration of gravity and friction.

(2) In case of plus move (up) and deceleration
Because torque of Gravity and friction work forward to the output torque of motor, the torque for acceleration/deceleration is as follows.

Maximum torque	:	130(=100+20+10) (Nm)	Speed 0 to 2000(min^{-1})
Torque at rapid traverse	:	109(=79+20+10) (Nm)	Speed 3000(min^{-1})
Minimum torque	:	88(=58+20+10) (Nm)	Speed 4000(min^{-1})

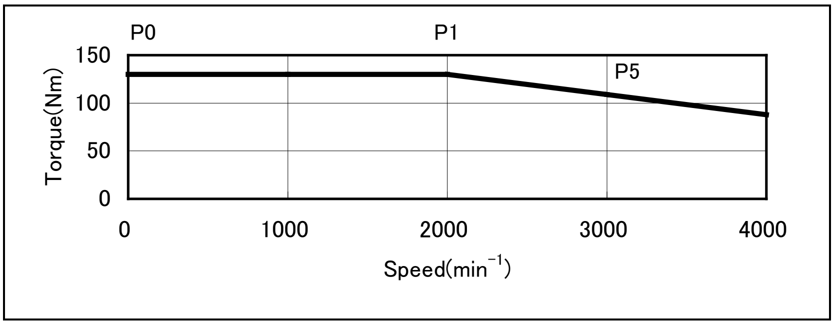


Fig. 7.2.5 (i) Torque for Acc/Dec in case of + move and deceleration with consideration of gravity and friction.

Parameter setting is as follows,

Fig.7.2.5 (e)

	Parameter number	Setting	Unit	Remarks
Acceleration at P0	19557	13514	0.01%	Set half the acceleration at P1, or 13514.
Acceleration at P1-P2	19558, 19559	27027	0.01%	At P1 and P2, 130(Nm) can be used for the acceleration/ deceleration, so set the ratio of the acceleration at 130 (Nm) to the reference acceleration. (2.7027 = 11146 / 4124)
Acceleration at P3-P4	19560 to 19561	0	0.01%	0 is set because P3 to P4 are skipped.
Acceleration at P5	19562	22662	0.01%	At P5, 109(Nm) can be used for the acceleration/deceleration, so set the ratio of the acceleration at 109 (Nm) to the reference acceleration. (2.2662 = 9346 / 4124)

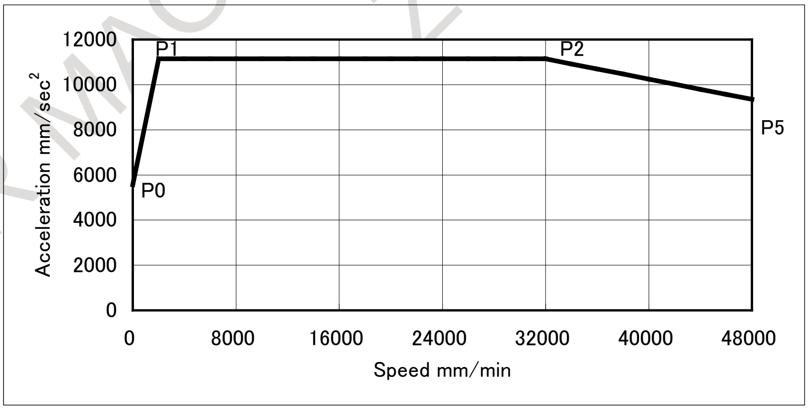


Fig. 7.2.5 (j) Acceleration pattern in case of + move and deceleration with consideration of gravity and friction.

(3) In case of minus move (down) and acceleration

Because torque of Gravity works forward to the output torque of motor and torque of friction works against the output torque of motor, torque for acceleration/deceleration is as follows.

Maximum torque	:	110(=100+20-10) (Nm)	Speed 0 to 2000(min^{-1})
Torque at rapid traverse	:	89(=79+20-10) (Nm)	Speed 3000(min^{-1})
Minimum torque	:	68(=58+20-10) (Nm)	Speed 4000(min^{-1})

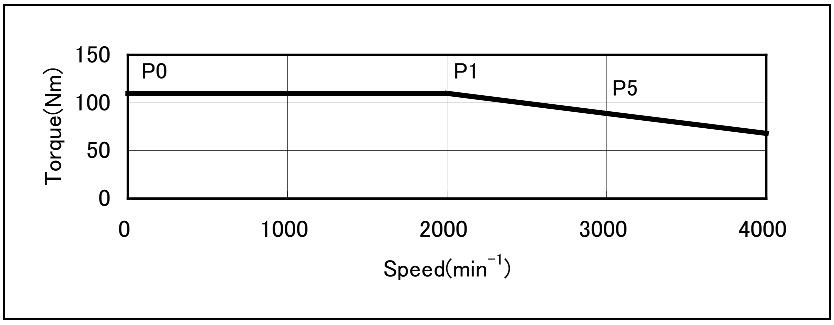


Fig. 7.2.5 (k) Torque for Acc/Dec in case of - move and acceleration with consideration of gravity and friction.

Parameter setting is as follows,

Fig.7.2.5 (f)

	Parameter number	Setting	Unit	Remarks
Acceleration at P0	19551	11435	0.01%	Set half the acceleration at P1, or 11435.
Acceleration at P1-P2	19552,19553	22869	0.01%	At P1 and P2, 110(Nm) can be used for the acceleration/ deceleration, so set the ratio of the acceleration at 110 (Nm) to the reference acceleration. (2.2869 = 9431 / 4124)
Acceleration at P3-P4	19554 to 19555	0	0.01%	0 is set because P3 to P4 are skipped.
Acceleration at P5	19556	18504	0.01%	At P5, 89(Nm) can be used for the acceleration/deceleration, so set the ratio of the acceleration at 89 (Nm) to the reference acceleration. (1.8504 = 7631 / 4124)

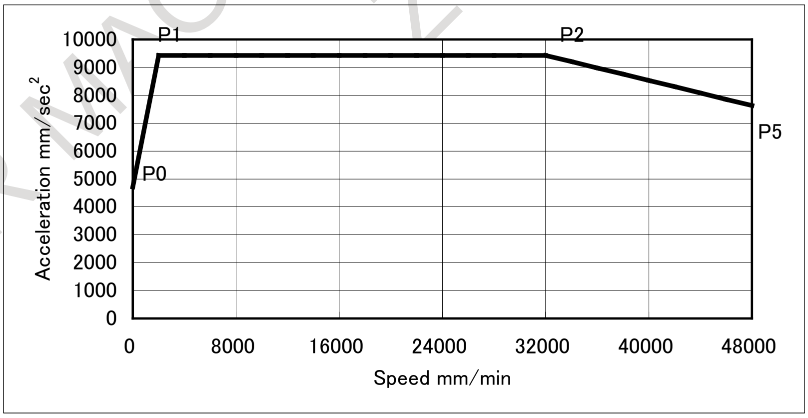


Fig. 7.2.5 (l) Acceleration pattern in case of - move and acceleration with consideration of gravity and friction.

(4) In case of minus move (down) and deceleration
Because torque of Gravity works against the output torque of motor and torque of friction works forward to the output torque of motor, torque for acceleration/deceleration is as follows.

Maximum torque	:	90(=100-20+10) (Nm)	Speed 0 to 2000(min^{-1})
Torque at rapid traverse	:	69(=79-20+10) (Nm)	Speed 3000(min^{-1})
Minimum torque	:	48(=58-20+10) (Nm)	Speed 4000(min^{-1})

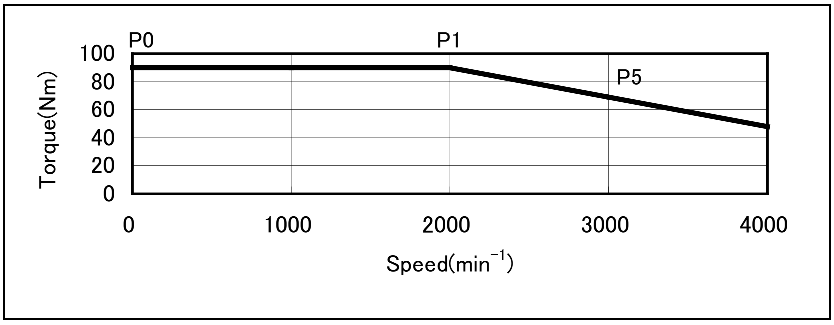


Fig. 7.2.5 (m) Torque for Acc/Dec in case of - move and deceleration with consideration of gravity and friction.

Parameter setting is as follows,

Fig.7.2.5 (g)

	Parameter number	Setting	Unit	Remarks
Acceleration at P0	19563	9356	0.01%	Set half the acceleration at P1, or 9356.
Acceleration at P1-P2	19564,19565	18712	0.01%	At P1 and P2, 90(Nm) can be used for the acceleration/ deceleration, so set the ratio of the acceleration at 90 (Nm) to the reference acceleration. (1.8712 = 7717 / 4124)
Acceleration at P3-P4	19566 to 19567,	0	0.01%	0 is set because P3 to P4 are skipped.
Acceleration at P5	19568	14345	0.01%	At P5, 69(Nm) can be used for the acceleration/deceleration, so set the ratio of the acceleration at 69 (Nm) to the reference acceleration. (1.4345 = 5916 / 4124)

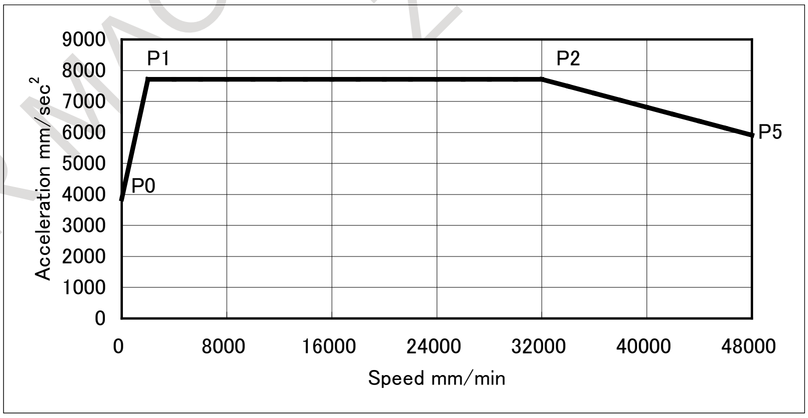


Fig. 7.2.5 (n) Acceleration pattern in case of - move and deceleration with consideration of gravity and friction.

Limitation

- Linear interpolation positioning

Optimum torque acceleration/deceleration is not enabled unless linear interpolation positioning is set (bit 1 (LRP) of parameter No. 1401 is set to 1).

- Valid modes and conditions

Optimum torque acceleration/deceleration is enabled when look-ahead acceleration/deceleration before interpolation (AI contour control mode) is active and the conditions under which look-ahead accel-

eration/deceleration before interpolation is performed are satisfied (bit 1 (LRP) of parameter No. 1401 is set to 1, and bit 5 (FRP) of parameter No. 19501 is set to 1, and parameter No.1671 is set to non-zero value).
However, when bit 0 (FAE) of parameter No. 11240 is set to 1, optimum torque acceleration/deceleration is enabled even if AI contour control is disabled.

- Target axes

Optimum torque acceleration/deceleration cannot be performed for a specific axis only. All axes operated by programmed commands are targeted for optimum torque acceleration/deceleration. This means that the PMC axes are excluded.

- Acceleration pattern

In the same direction move, it is necessary that the acceleration during deceleration is set 1/3 of one during acceleration at least.

Also, the acceleration pattern data must be set so that the time required for the deceleration from the rapid traverse rate to a speed of 0 does not exceed 4000 (msec). This does not include the acceleration change time of bell-shaped acceleration/deceleration.

If the deceleration acceleration ratio or the time required for the deceleration to a speed of 0 exceeds the above range, alarm DS1710, "ILLEGAL ACC. PARAMETER (OPTIMUM TORQUE ACC/DEC)" will be issued at the time rapid traverse is executed.

A slight error will occur between the specified acceleration and the actual acceleration.

Parameter

1671	Maximum allowable acceleration rate in acceleration/deceleration before interpolation for linear rapid traverse for each axis, or allowable reference acceleration rate in optimum torque acceleration/deceleration

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

(1) Set a maximum allowable acceleration rate in look-ahead acceleration/ deceleration before interpolation for linear rapid traverse.
If a value greater than 100000.0, the value is clamped to 100000.0.
If 0 is set, the specification of the following is assumed:
1000.0 mm/sec^2
100.0 inch/sec^2
100.0 degrees/sec^2
If 0 is specified for all axes, however, look-ahead acceleration/deceleration before interpolation is not performed.

(2) Allowable reference acceleration rate in optimum torque acceleration/deceleration

1672	Acceleration change time of bell-shaped acceleration/deceleration before interpolation for linear rapid traverse, or acceleration change time of bell-shaped acceleration/deceleration in optimum torque acceleration/deceleration

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 200

(1) Set an acceleration change time of bell-shaped acceleration/ deceleration for linear rapid traverse (time for changing from the state of constant feedrate (A) to the state of constant acceleration/deceleration (C) at the acceleration rate calculated from the acceleration rate set in parameter No. 1671: time of (B) in the figure below).

(2) Set an acceleration change time of bell-shaped acceleration/ deceleration in optimum torque acceleration/deceleration (time for changing from the state of constant feedrate (A) to the state of acceleration/deceleration (C) at the acceleration rate calculated from optimum torque acceleration/deceleration: time of (B) in the figure).

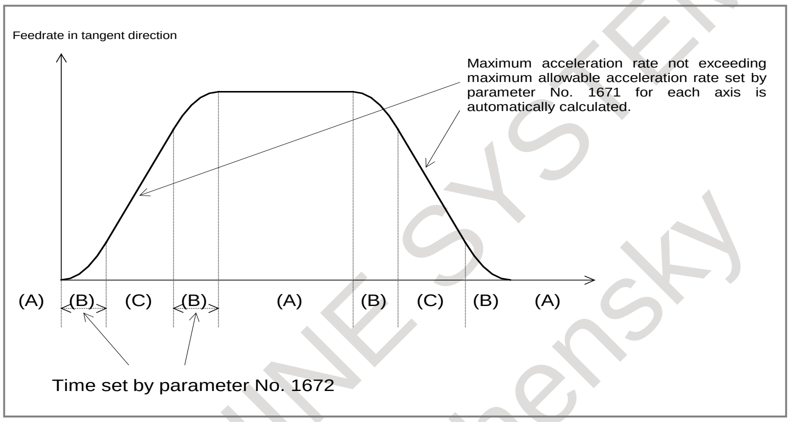


Fig. 7.2.5 (o)

	#7	#6	#5	#4	#3	#2	#1	#0
11240								FAE

[Input type] Parameter input
[Data type] Bit path

#0 FAE During positioning when the AI contour control mode is canceled, the optimum torque acceleration/deceleration is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
19501			FRP					

[Input type] Parameter input
[Data type] Bit path

#5 FRP Linear rapid traverse is:
0: Acceleration/deceleration after interpolation
1: Look-ahead acceleration/deceleration before interpolation

Set a maximum allowable acceleration rate for each axis in parameter No. 1671.
When using look-ahead bell-shaped acceleration/deceleration before interpolation, set an acceleration rate change time in parameter No. 1672.

When this parameter is set to 1, look-ahead acceleration/deceleration before interpolation is also applied to rapid traverse if all conditions below are satisfied. At this time, acceleration/deceleration after interpolation is not applied.

- Bit 1 (LRP) of parameter No. 1401 is set to 1: Linear interpolation type positioning
- A value other than 0 is set in parameter No. 1671 for an axis.
- The AI contour control mode is set.

If all of these conditions are not satisfied, acceleration/deceleration after interpolation is applied.

	#7	#6	#5	#4	#3	#2	#1	#0
19540								FAP

[Input type] Parameter input
[Data type] Bit path

#0 FAP Optimal torque acceleration/deceleration is:
0: Disabled.
1: Enabled.

When the linear positioning parameters, namely bit 1 (LRP) of parameter No. 1401 and bit 0 (FAP) of parameter No. 19540, are set to 1, and a value other than 0 is set in reference acceleration parameter No. 1671 for an axis, the acceleration/deceleration for rapid traverse becomes optimal torque acceleration/deceleration in the mode for look-ahead acceleration/deceleration before interpolation (or the AI contour control mode). Optimal torque acceleration/ deceleration is controlled according to parameter-set restricted acceleration curve data.

Setting an acceleration pattern

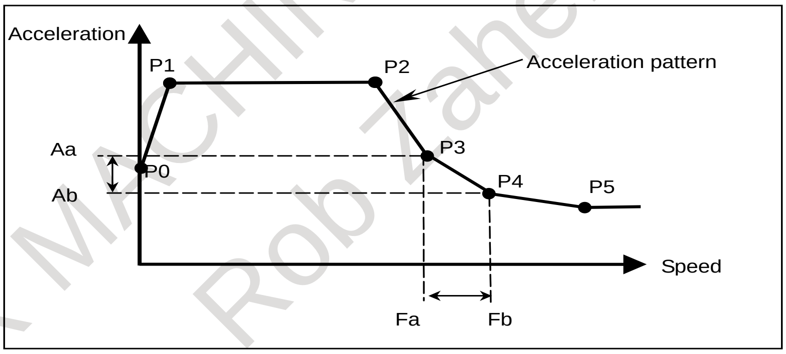


Fig. 7.2.5 (p)

Set the speed at each of the acceleration setting points (P0 to P5) in a corresponding parameter, then in parameters for each axis, set acceleration rates applicable in the following four cases at these speeds: when a movement in the positive direction is accelerated, when a movement in the positive direction is decelerated, when a movement in the negative direction is accelerated, and when a movement in the negative direction is decelerated.
The line connecting the acceleration setting points is the acceleration pattern.
The acceleration rate for each axis is calculated. For example, between speeds Fa to Fb in the above figure, the acceleration rates corresponding to these speeds, Aa to Ab, are used for calculation.
The tangent acceleration is controlled so that it does not exceed the calculated acceleration rate for each axis.

> ⚠ **CAUTION**
> When an acceleration pattern is set, setting a high acceleration rate immediately after a speed of 0 can cause an impact on the machine, so it is not desirable. Therefore, **be sure to apply a relatively low acceleration rate at a speed of 0** as shown in the above figure.

19541	**Optimal torque acceleration/deceleration (speed at P1)**
to	**to**
19544	**Optimal torque acceleration/deceleration (speed at P4)**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01%
[Valid data range] 0 to 10000

The speeds at acceleration setting points P1 to P4 are to be set with speed parameters Nos. 19541 to 19544 as ratios to the rapid traverse speed (parameter No. 1420). The speed at P0 is 0, and the speed at P5 is the rapid traverse rate specified with parameter No. 1420. Any acceleration setting point for which the speed parameter (one of Nos. 19541 to 19544) is set to 0 will be skipped.

19545	**Optimal torque acceleration/deceleration (acceleration at P0 during movement in + direction and acceleration)**
to	**to**
19550	**Optimal torque acceleration/deceleration (acceleration at P5 during movement in + direction and acceleration)**
19551	**Optimal torque acceleration/deceleration (acceleration at P0 during movement in - direction and acceleration)**
to	**to**
19556	**Optimal torque acceleration/deceleration (acceleration at P5 during movement in - direction and acceleration)**
19557	**Optimal torque acceleration/deceleration (acceleration at P0 during movement in + direction and deceleration)**
to	**to**
19562	**Optimal torque acceleration/deceleration (acceleration at P5 during movement in + direction and deceleration)**
19563	**Optimal torque acceleration/deceleration (acceleration at P0 during movement in - direction and deceleration)**
to	**to**
19568	**Optimal torque acceleration/deceleration (acceleration at P5 during movement in - direction and deceleration)**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01%
[Valid data range] 0 to 32767

For each travel direction and each acceleration/deceleration operation, set the allowable acceleration rate at each of the acceleration setting points (P0 to P5). As an allowable acceleration rate, set a ratio to the value set in the reference acceleration parameter No. 1671. When 0 is set, the specification of 100% is assumed.

Alarm and message

Number	Message	Description
DS1710	ILLEGAL ACC. PARAMETER (OPTIMUM TORQUE ACC/DEC)	There are errors in the parameters of permissible acceleration for Optimum Torque Acceleration/Deceleration. One of the following is the cause. 1) The ratio of the acceleration for deceleration to the acceleration for the acceleration is lower than the limited value. 2) The time to decelerate to 0 is larger than the maximum.

Reference item

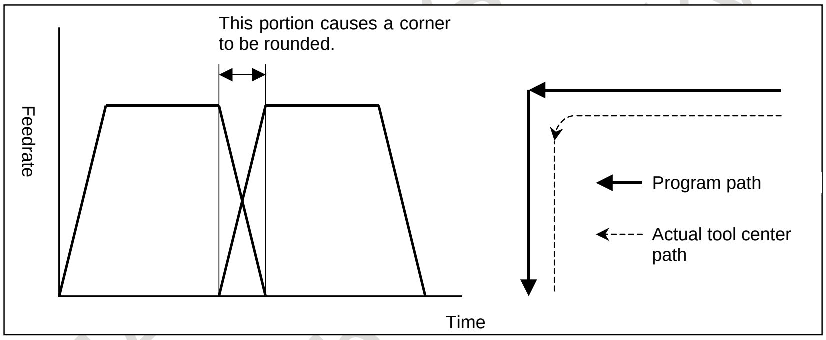
Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Optimum torque acceleration/deceleration

7.2.6 Corner Control

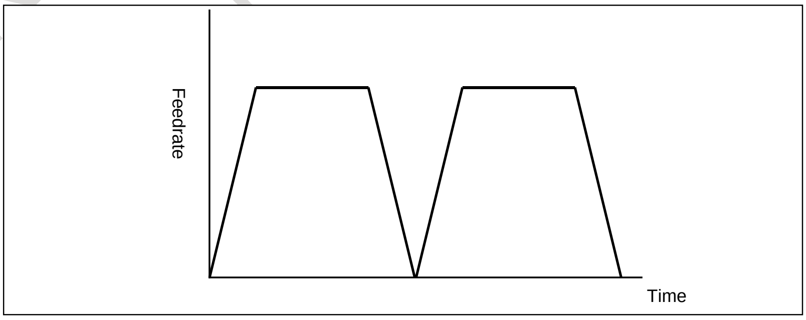
7.2.6.1 In-position check signal

Overview

On general CNCs, the feedrate during cutting feed never becomes 0 between two successive blocks. So, a corner may be rounded.



By using the in-position check signal, control can be exercised so that machining can proceed to the next block after checking that the acceleration/deceleration processing of the previous block is completed and the effective area has been entered.



Signal

In-position check signal SMZ<Gn053.6>

[Classification] Input signal

[Function] This signal selects whether to make an in-position check between successive cutting feed blocks during automatic operation.
In general, slight rounding occurs between successive cutting feed blocks because of an acceleration/deceleration delay or servo delay.
By making an in-position check (to proceed to the next block after waiting until acceleration/deceleration and servo delays disappear), this rounding can be eliminated.

[Operation] When this signal is set to "0", machining proceeds to the next block without making an in-position check between successive cutting feed blocks during automatic operation.
When this signal is set to "1", machining proceeds to the next block after making an in-position check between successive cutting feed blocks during automatic operation.

NOTE
Before and after a block for positioning, an in-position check is made, regardless of the setting of this signal.
(However, the setting of bit 5 (NCI) of parameter No. 1601 is followed.)

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn053		SMZ						

7.2.6.2 In-position check

Overview

Whether the position of the servo motor is within a specified range is checked.
If the in-position check function is enabled, the CNC checks the position during deceleration. If the position is found to exceed the specified range, the CNC does not execute the next block.

NOTE
1 The purpose of in-position check is to check that the servo motor has reached within a specified range (specified with parameter No. 1826).
2 In-position check is executed for all axes in the path. Therefore, the axis which is not commanded is also the target of in-position check.

Signal

In-position signals INP1 to INP8<Fn104>

[Classification] Output signal

[Function] These signals indicate that the control axes are set to the in-position condition.
They are provided for each control axis, and the end number in the signal name corresponds to the control axis number.

INPx

x : 1 ..... The 1st axis is set to the in-position condition.
2 ..... The 2nd axis is set to the in-position condition.
3 ..... The 3rd axis is set to the in-position condition.
: :

[Output cond.] These signals turn to "1" in the following case :

- The move command of the corresponding control axis is not specified and the servo error of the corresponding control axis is within the specified allowance.

> **NOTE**
> Even if the servo error of the corresponding control axis is within the specified allowance (in-position width), the signal is not set to "1" when the axis is moving or there is acceleration/deceleration delay.

These signals turn to "0" in the following cases :

- The move command of the corresponding control axis is specified.
- The servo error of the corresponding control axis exceeds the specified allowance.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn104	INP8	INP7	INP6	INP5	INP4	INP3	INP2	INP1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601			NCI					

[Input type] Parameter input
[Data type] Bit path

#5 NCI An in-position check:

0: Confirms that the specified feedrate becomes 0 (the acceleration/deceleration delay becomes 0) at deceleration time and that the machine position has reached a specified position (the servo positional deviation is within the in-position width set by parameter No. 1826).

1: Confirms only that the specified feedrate becomes 0 (the acceleration/deceleration delay becomes 0) at deceleration time.

1826	In-position width for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

The in-position width is set for each axis.
When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position. (The machine is in the in-position state.)

Note

> **NOTE**
> 1 The in-position signals INP1 to INP8 <Fn104> may turn to "1" even during the movement if the axis is fed at very low speed.
> 2 The in-position check function is enabled at the interface between two cutting blocks, when the exact stop command (G09) or exact stop mode command (G61) is specified

7.2.6.3 In-position check disable signal

Overview

In-position check disable signal NOINPS <Gn023.5> can be used to specify whether an in-position check is executed.

Explanation

In-position check disable signal NOINPS <Gn023.5> can be used to disable a in-position check between blocks. When an in-position check is disabled, the CNC does not perform an in-position check during deceleration. That is, the CNC starts the next block before entering the in-position. This reduces the cycle time.

NOTE

1. When an in-position check is disabled, the CNC starts the next block before entering the in-position. Note that this may cause a difference between the program command and the actual tool path.
2. In-position check disable signal NOINPS <Gn023.5> can disable an in-position check between blocks, but cannot disable an in-position check by the following G codes.
   - Reference position return (G28)
   - 2nd/3rd/4th reference position return (G30)
   - Machine coordinate system selection (G53)
   - Single direction positioning (G60)

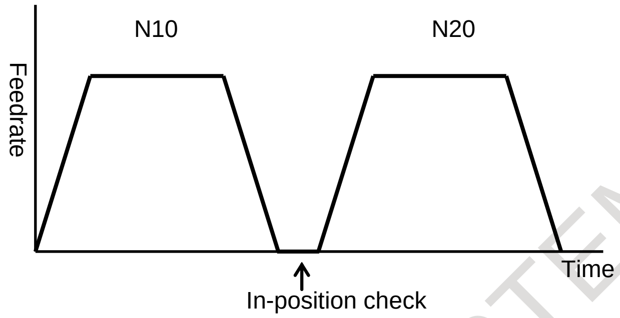
   However, this signal can disable an in-position check at the midpoint of G28 and G30, or at the motion end point of G60.
   To disable an in-position check in the reference position in G28 and G30, specify reference position return (G28.2, G30.2).

Example

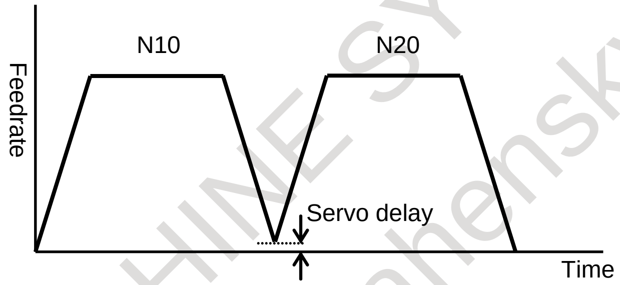
```
O0001;
N10 G00 X50.0;
N20 G00 X100.0;
N30 M30;
```

Note) The program starts at the position in which X-coordinate is 0.0.

<1> When NOINPS <Gn023.5> is "0"
(An in-position check is performed.)



<2> When NOINPS <Gn023.5> is "1"
(An in-position check is not performed.)



Signal

In-position check disable signal NOINPS <Gn023.5>

[Classification] Input signal

[Function] Specifies whether an in-position check is performed.

[Operation] When this signal is set to "1", the controller operates as follows.

An in-position check is not performed regardless of the setting of bit 5 (NCI) of parameter No. 1601. That is, the next block is executed before the in-position is entered.

The operation when an in-position check is not performed is the same as that in the case where bit 5 (NCI) of parameter No. 1601 is 1.

When this signal is set to "0", the controller operates as follows.

An in-position check follows the setting of bit 5 (NCI) of parameter No. 1601.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn023			NOINPS					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601			NCI					

[Input type] Parameter input
[Data type] Bit path

#5 NCI An in-position check:
0: Confirms that the specified feedrate becomes 0 (the acceleration/deceleration delay becomes 0) at deceleration time and that the machine position has reached a specified position (the servo positional deviation is within the in-position width set by parameter No. 1826).
1: Confirms only that the specified feedrate becomes 0 (the acceleration/deceleration delay becomes 0) at deceleration time.

7.2.6.4 In-position check independently of feed/rapid traverse

Overview

If separate in-position check for cutting feed and rapid traverse is executed, a small in-position check range can be specified between those cutting feed blocks that require a high degree of precision. A large in-position check range can be specified between those rapid traverse blocks that require quick positioning.

Signal

See Item “In-position check.”.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601			NCI					

[Input type] Parameter input
[Data type] Bit path

#5 NCI An in-position check:
0: Confirms that the specified feedrate becomes 0 (the acceleration/deceleration delay becomes 0) at deceleration time and that the machine position has reached a specified position (the servo positional deviation is within the in-position width set by parameter No. 1826).
1: Confirms only that the specified feedrate becomes 0 (the acceleration/deceleration delay becomes 0) at deceleration time.

	#7	#6	#5	#4	#3	#2	#1	#0
1801			CIN	CCI				

[Input type] Parameter input
[Data type] Bit path

#4 CCI As the in-position width for cutting feed:
0: The parameter No. 1826 applicable to rapid traverse as well is used.
1: The parameter No. 1827 dedicated to cutting feed is used.
This parameter enables the in-position width for cutting feed (parameter No. 1827) to be set instead of the in-position width for rapid traverse (parameter No. 1826).

By setting bit 4 (CCI) of parameter No. 1801, choose whether to use this function or the conventional in-position check function.
This function, when specified, is enabled for all axes. So, for an axis that does not require this function, set the same data in parameters Nos. 1826 and 1827.

#5 CIN When CCI is set to 1, the dedicated parameter for specifying an in-position width for cutting feed is used:
0: Only when the next block specifies cutting feed.
1: Regardless of the next block.
The table indicates the relationships between the parameters for cutting feed and rapid traverse.

		Bit 5 (CIN) of parameter No. 1801			
		0		**1**	
Bit 4 (CCI) of parameter No. 1801	**0**	Rapid traverse → Rapid traverse	No. 1826	Rapid traverse → Rapid traverse	No. 1826
		Rapid traverse → Cutting feed	No. 1826	Rapid traverse → Cutting feed	No. 1826
		Cutting feed → Cutting feed	No. 1826	Cutting feed → Cutting feed	No. 1826
		Cutting feed → Rapid traverse	No. 1826	Cutting feed → Rapid traverse	No. 1826
	1	Rapid traverse → Rapid traverse	No. 1826	Rapid traverse → Rapid traverse	No. 1826
		Rapid traverse → Cutting feed	No. 1826	Rapid traverse → Cutting feed	No. 1826
		Cutting feed → Cutting feed	No. 1827	Cutting feed → Cutting feed	No. 1827
		Cutting feed → Rapid traverse	No. 1826	Cutting feed → Rapid traverse	No. 1827

The parameters CCI and CIN can also be applied to a Cs axis.

1826	In-position width for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

The in-position width is set for each axis.
When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position. (The machine is in the in-position state.)

1827	In-position width in cutting feed for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set an in-position width for each axis in cutting feed. This parameter is used when bit 4 (CCI) of parameter No. 1801=1.

Note

> **NOTE**
> If the bit 5 (NCI) of parameter No. 1601 is set to 1, so that position check is not performed during deceleration, this function is invalid. The system starts execution of the next block as soon as deceleration has been completed, without checking whether the servo position error is within the specified range.

7.2.7 Feed Forward in Rapid Traverse

Overview

Feed-forward control can be available even during rapid traverse. In this case, the servo position deviation is reduced, thus it reduces the time required for positioning to within the specified range.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1800					FFR			

[Input type] Parameter input
[Data type] Bit path

#3 FFR Feed-forward control in rapid traverse is:
0: Disabled
1: Enabled
Feed-forward is enabled only in normal cutting feed. When this parameter is set to 1, feed-forward is enabled in rapid traverse as well. This capability reduces the servo positional deviation, thus reducing the time required to enter the in-position width at the time of positioning.

> **NOTE**
> The parameter setting becomes valid after reference position return is completed.

Reference item

For details of this function, refer to the "FANUC SERVO MOTOR α*i* series Maintenance Manual (B-65285EN)."

7.2.8 Optimum Acceleration/Deceleration for Rigid Tapping

Overview

This function can be used to flexibly set the acceleration/deceleration during cutting in rigid tapping according to the torque characteristics of a spindle motor and the mechanical characteristics such as machine friction. Depending on the torque characteristics of a spindle motor and the mechanical characteristics, the acceleration/deceleration performance (referred to below as the maximum acceleration curve) that can be output is not symmetrical in the low-speed and high-speed parts. The conventional acceleration/deceleration (linear-shaped/bell-shaped) is symmetrical, so it was impossible to make the most of the motor performance.
This function can be used to perform acceleration/deceleration so that the actual speed curve can follow the maximum acceleration curve as close as possible. This can make the most of the motor performance and reduce the cutting time. When rigid tapping is used in an area where the acceleration of the spindle motor is constant, however, reduction in the cutting time cannot be expected.

If the acceleration pattern is set for the parameter for each gear, rigid tapping by acceleration following the maximum acceleration curve.

In acceleration/deceleration during withdrawal, the acceleration/deceleration pattern during cutting is used.

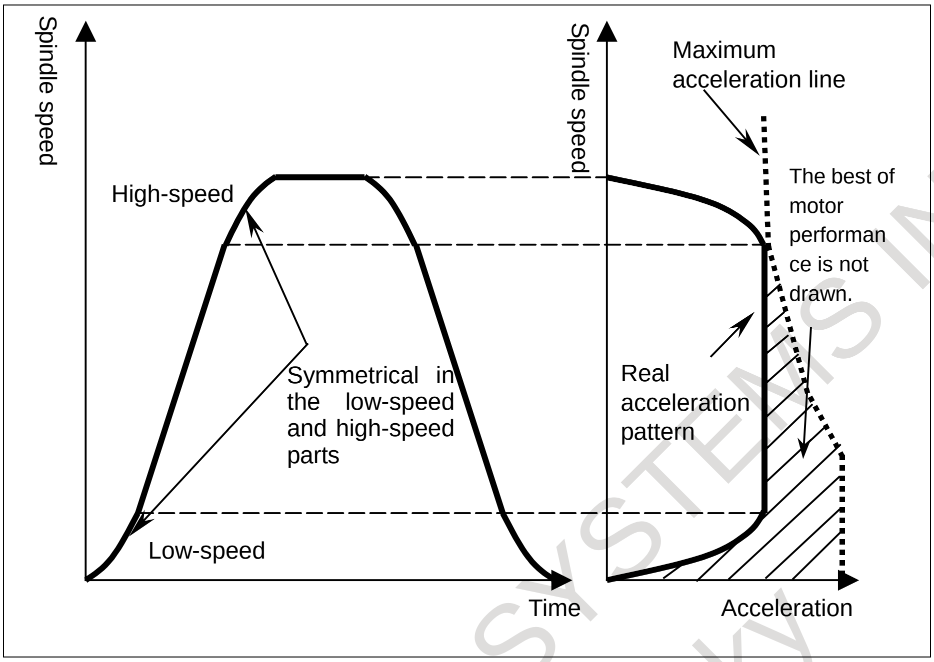


Fig. 7.2.8 (a) Conventional acceleration/deceleration (bell-shaped)

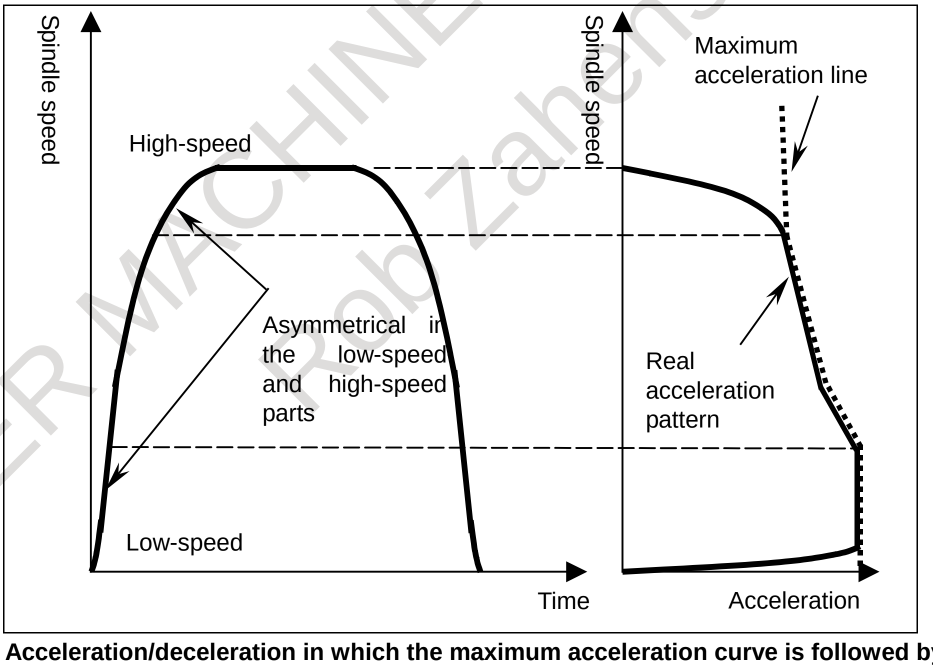


Fig. 7.2.8 (b) Acceleration/deceleration in which the maximum acceleration curve is followed by the actual acceleration curve by this function

Explanation

This function selects an acceleration pattern set in the parameter based on the acceleration/deceleration state of the spindle and the selected gears (between the spindle and the spindle motor), calculates the allowable acceleration of the spindle from the current spindle acceleration, and performs rigid tapping by controlling acceleration/deceleration so that the allowable acceleration of the spindle is not exceeded.

NOTE

In the text in this section, up to four stages of gears between the spindle and the spindle motor in rigid tapping are described. Up to four stages of gears are used in lathe systems, but up to three stages of gears are used in machining centers.

- Setting optimum acceleration/deceleration

Set bit 0 (RAU) of parameter No. 11420 to 1 to enable this function.

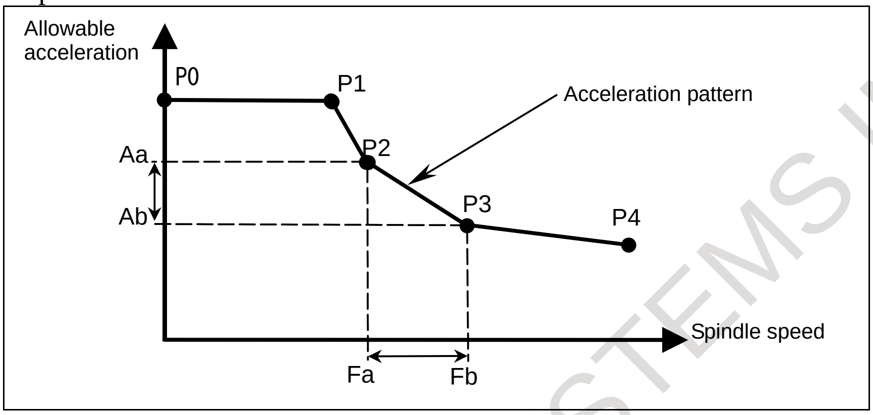


Fig. 7.2.8 (c) Acceleration setting point

For the parameter for each gear, set the rotation speed and allowable acceleration in the acceleration setting points (P0 to P4) for each spindle acceleration/deceleration. The kinked line connecting the acceleration setting points is the acceleration pattern.
In changes in the allowable acceleration, when, for example, speed Fa shifts to speed Fb in Fig. 7.2.8 (c), the allowable accelerations Aa and Ab corresponding to these speeds are used.

Table 7.2.8 (a) Parameters for acceleration patterns for each gear

Gear	Parameter number for maximum acceleration	Parameter number for maximum spindle speed	Acceleration setting points	Parameter number for spindle speed	Allowable acceleration parameter number Forward/Reverse	
					Acceleration	Deceleration
Gear 1	11421	5241	P0	None (0%)	11441	11461
			P1	11429	11442	11462
			P2	11430	11443	11463
			P3	11431	11444	11464
			P4	None (100%)	11445	11465
Gear 2	11422	5242	P0	None (0%)	11446	11466
			P1	11432	11447	11467
			P2	11433	11448	11468
			P3	11434	11449	11469
			P4	None (100%)	11450	11470
Gear 3	11423	5243	P0	None (0%)	11451	11471
			P1	11435	11452	11472
			P2	11436	11453	11473
			P3	11437	11454	11474
			P4	None (100%)	11455	11475
Gear 4	11424	5244	P0	None (0%)	11456	11476
			P1	11438	11457	11477
			P2	11439	11458	11478
			P3	11440	11459	11479
			P4	None (100%)	11460	11480

- Set the parameter of the gear to be used.
- Set parameters Nos. 11421 to 11424 to the maximum acceleration (rev/sec^2).
  If these parameters are set to 0, a DS alarm DS1711, "ILLEGAL ACC. PARAMETER (RIGID TAPPING OPTIMUM ACC/DEC)" is issued during execution.
- Since the spindle speed for P0 does not need to be set because it is 0%.
- Since the spindle speed for P4 does not need to be set because it is 100%.
- Set parameters Nos. 11429 to 11440 to the spindle speed for P1 to P3 as the ratio relative to the maximum spindle speed (parameters Nos. 5241 to 5244) in rigid tapping.
- Set parameters Nos. 11441 to 11480 to the allowable acceleration for P0 to P4 as the ratio relative to the maximum acceleration (parameters Nos. 11421 to 11424).
- Skip the acceleration setting point in which parameters Nos. 11429 to 11440 for spindle speed are set to 0 and treat a line connecting to the next acceleration setting point in which a nonzero value is set.
- When parameters Nos. 11441 to 11480 for allowable acceleration are set to 0, a value of 100% is assumed.

- Expression indicating the relationship between torque and acceleration

When the torque is $T(N \cdot m)$ and the inertia is $J\left(kg \cdot m^2\right)$, acceleration $A\left(rev/\sec^2\right)$ is calculated by the following expression.

$$\begin{aligned} A &= T(N \cdot m) / J\left(kg \cdot m^2\right)/(2\pi) \\ &= T\left(\left(kg \cdot m/\sec^2\right)(m)\right)/J\left(kg \cdot m^2\right)/(2\pi) \\ &= T/\left((2\pi) \cdot J\right) \end{aligned}$$

- Example of setting an acceleration pattern

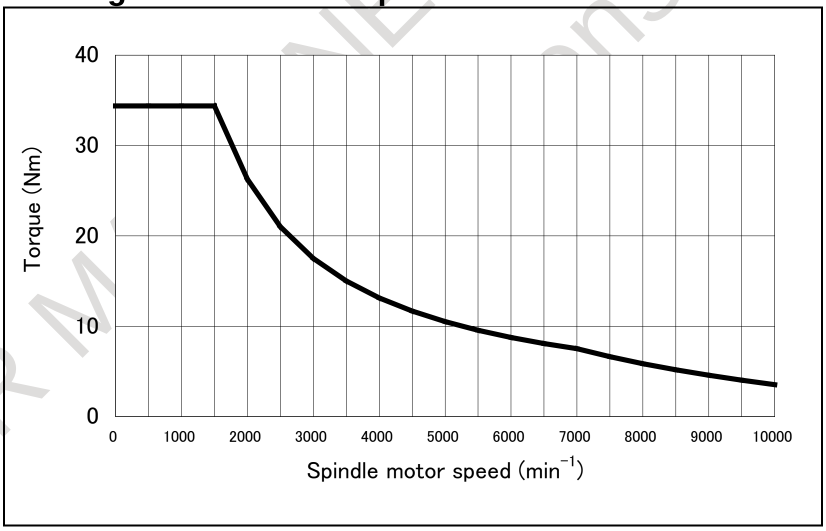


Fig. 7.2.8 (d) Speed vs. torque characteristics of model αiI3/10000-B in the operational area of 30 minutes and S3 60%

Structure
- The spindle and spindle motor are mutually connected directly.
- The spindle inertia is almost the same as the rotor inertia.
- The total inertial is double the rotor inertia.
- Model αiI3/10000-B motor

Specification of the model αiI3/10000-B motor

Rotor inertia	0.0148 $\left(kg \cdot m^2\right)$
Maximum torque	34.3 $(N \cdot m)$: Speed 0 to 1500 $\left(\min^{-1}\right)$
Spindle inertia	0.0148 $\left(kg \cdot m^2\right)$

The acceleration $A(rev/\sec^2)$ at the maximum torque of $34.3(N \cdot m)$ is calculated from the above expression, which indicates the relationship between torque vs. acceleration.

$$A = 34.3/((2\pi) \cdot 2.0 \cdot 0.0148)(rev/\sec^2)$$
$$= 184.426(rev/\sec^2)$$

An example of setting the parameters for acceleration patterns is shown in Table 7.2.8 (b).
Setting conditions

- The acceleration is constant for both acceleration and deceleration.
- The spindle and spindle motor is mutually connected directly (1:1).
- The maximum spindle speed is 6000 min^{-1}.

Table 7.2.8 (b) Example of setting the parameters for acceleration patterns

	Parameter No.	Setting value (unit)	Remarks
Maximum speed of spindle	5241	6000 (min^{-1})	
Maximum acceleration	11421	184 (rev/sec^2)	Sets the maximum acceleration A.
Spindle speed (P1)	11429	25 (%)	Since the rotation speed for a torque of 34.3 (Nm) is up to 1500 (min^{-1}), set P1 to 25% (1500 min^{-1}).
Spindle speed (P2)	11430	33 (%)	Since the torque drops linearly from 1500 (min^{-1}) to 2000 (min^{-1}), set P2 to 33% (1980 min^{-1}).
Spindle speed (P3)	11431	50 (%)	Since the torque drops linearly from 2000 (min^{-1}) to 3000 (min^{-1}), set P3 to 50% (3000 min^{-1}).
Acceleration (P0)	11441 11461	100 (%)	Set the acceleration at a rotation speed of 0 to 100% (torque 34.3 (Nm)).
Acceleration (P1)	11442 11462	100 (%)	Set the acceleration at the rotation speed of P1 to 100% (torque 34.3 (Nm)).
Acceleration (P2)	11443 11463	72 (%)	Set the acceleration at the rotation speed of P2 to 72% (torque 25 (Nm)).
Acceleration (P3)	11444 11464	43 (%)	Set the acceleration at the rotation speed of P3 to 43% (torque 15 (Nm)).
Acceleration (P4)	11445 11465	23 (%)	Set the acceleration at the rotation speed of P4 to 23% (torque 8 (Nm)).

If the parameters are set as described in Table 7.2.8 (c), the acceleration pattern is assumed as shown in Fig. 7.2.8 (e).

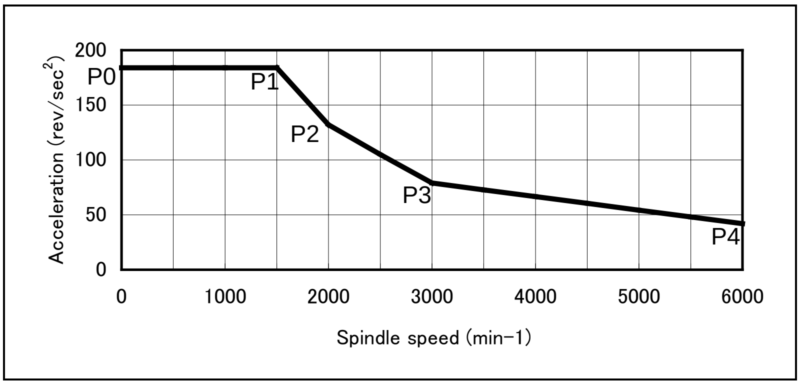


Fig. 7.2.8 (e)

At P0, the rotation speed is 0 min^{-1} and the acceleration is 184 rev/sec^2.
At P1, the rotation speed is 1500 min^{-1} and the acceleration is 184 rev/sec^2.

At P2, the rotation speed is 1980 min^{-1} and the acceleration is 132 rev/sec^2.
At P3, the rotation speed is 3000 min^{-1} and the acceleration is 79 rev/sec^2.
At P4, the rotation speed is 6000 min^{-1} and the acceleration is 42 rev/sec^2.

NOTE

1. The acceleration pattern needs to be set according to the specification of a spindle motor and the characteristics of a machine.
2. Maximum spindle speed parameters Nos. 5241 to 5244 and spindle speed parameters Nos. 11429 to 11440 may not match the speed of the spindle motor. The acceleration pattern needs to be set by taking the actual speed of the spindle motor into account.
   For example, when half the spindle speed is the speed of the spindle motor, the acceleration at the spindle speed of 6000 min^{-1} needs to be determined on the basis of the torque at the motor speed of 3000 min^{-1}.
3. The acceleration pattern needs to be changed when optimum acceleration/deceleration for rigid tapping is not operating in any path.

- Bell-shaped acceleration/deceleration

When using bell-shaped acceleration/deceleration, set parameters Nos. 11425 to 11428 to the bell-shaped acceleration change time.
When not using bell-shaped acceleration/deceleration, set these parameters to 0.

- Acceleration/deceleration after interpolation

When optimum acceleration/deceleration for rigid tapping is enabled, acceleration/deceleration after compensation is not applied.

- Override

Various types of override are disabled, but the following override can be enabled.
- Extraction override
- Override signal

See the Section 11.12, “Rigid Tapping” for details.

- Retraction for rigid tapping

Even in retraction for rigid tapping, optimum acceleration/deceleration for rigid tapping is enabled.

- Functions that can be used simultaneously

Optimum acceleration/deceleration for rigid tapping can be used concurrently with the following functions.
- 3-dimensional coordinate system conversion (3-dimensional rigid tapping)
- Tilted working plane indexing
- Axis synchronous control
- Composite control
- Tandem control
- Multi-spindle control
- Spindle command synchronous control

Restrictions

- Rigid tapping types

Optimum acceleration/deceleration for rigid tapping cannot be used in the following rigid tapping functions.
- Rigid tapping with analog spindle
- Rigid tapping by manual handle
- Rigid tapping with Cs contour control

- Rigid tapping with servo motor

- Functions that cannot be used simultaneously

Optimum acceleration/deceleration for rigid tapping cannot be used concurrently with the following functions.

- General purpose retract
  Retract signal RTRCT <Gn066.4> is ignored while the optimum acceleration/deceleration for rigid tapping is executed.

- Acceleration line pattern

- An acceleration pattern can be set for both acceleration and deceleration, but the ratio of deceleration to acceleration must be one-third or greater. For example, when the acceleration of a certain acceleration setting point (Pn) is set to 90%, the deceleration of corresponding to the point must be 30% or greater.
- The acceleration pattern of deceleration needs to be set so that the time required for deceleration from the maximum spindle speed to a rotation speed of 0 is 4000 msec or less. However, this time does not include acceleration change time for bell-shaped acceleration/deceleration.
- If the ratio of deceleration to acceleration or the time required for deceleration to a rotation speed of 0 falls outside the range, an alarm DS1711, “ILLEGAL ACC. PARAMETER (RIGID TAPPING OPTIMUM ACC/DEC)” is issued during execution.
- There is slight calculation error between the set acceleration and the actual acceleration.

Example of the adjustment (Tuning assist by SERVO GUIDE)

“Tuning assist of rigid tapping acc./dec.” in SERVO GUIDE is used for tuning.
“Tuning assist of rigid tapping acc./dec.” is supported in SERVO GUIDE edition 9.40 or later.

(1)Measurement of spindle speed waveform

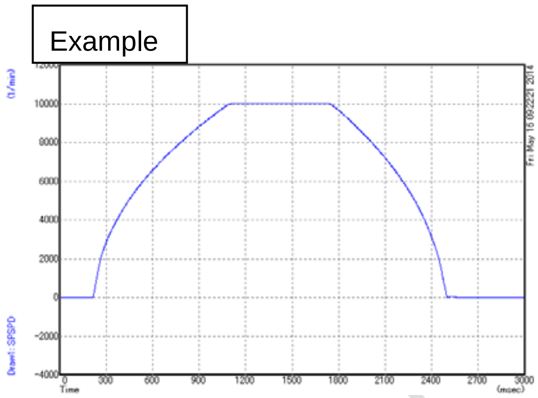
Measure Acc./Dec. waveform of spindle speed (SPSPD) to maximum spindle speed on velocity control mode.
Be careful about the following points then so that you can get the similar characteristic in the rigid tapping mode. (But, when a synchronous spindle motor is used, these items are not to be cared.)

When flux control is applied (bit 2 of parameter No.4002 = 1), set bit 2 of parameter No.4007 to 1 before measurement as the temporary setting only for the measurement. Reset bit 2 of parameter No.4007 to 0 after the measurement.
Change the motor voltage of velocity control mode (No.4083 etc.) to 100 temporarily.
After motor excitation command is input, start the measurement after waiting for a while (0.5~2.0 seconds or so) before excitation rising.

> **NOTE**
> If the parameter table for each motor model has setting “bit 2 of parameter No.4007 = 1”, it isn’t necessary to reset bit 2 of parameter No.4007 to 0 after the measurement. Please refer to FANUC AC SPINDLE MOTOR α*i*/β*i* series BUILT-IN SPINDLE MOTOR B*i* series parameter manual (B-65280EN) about the detail of the parameter table for each motor model.

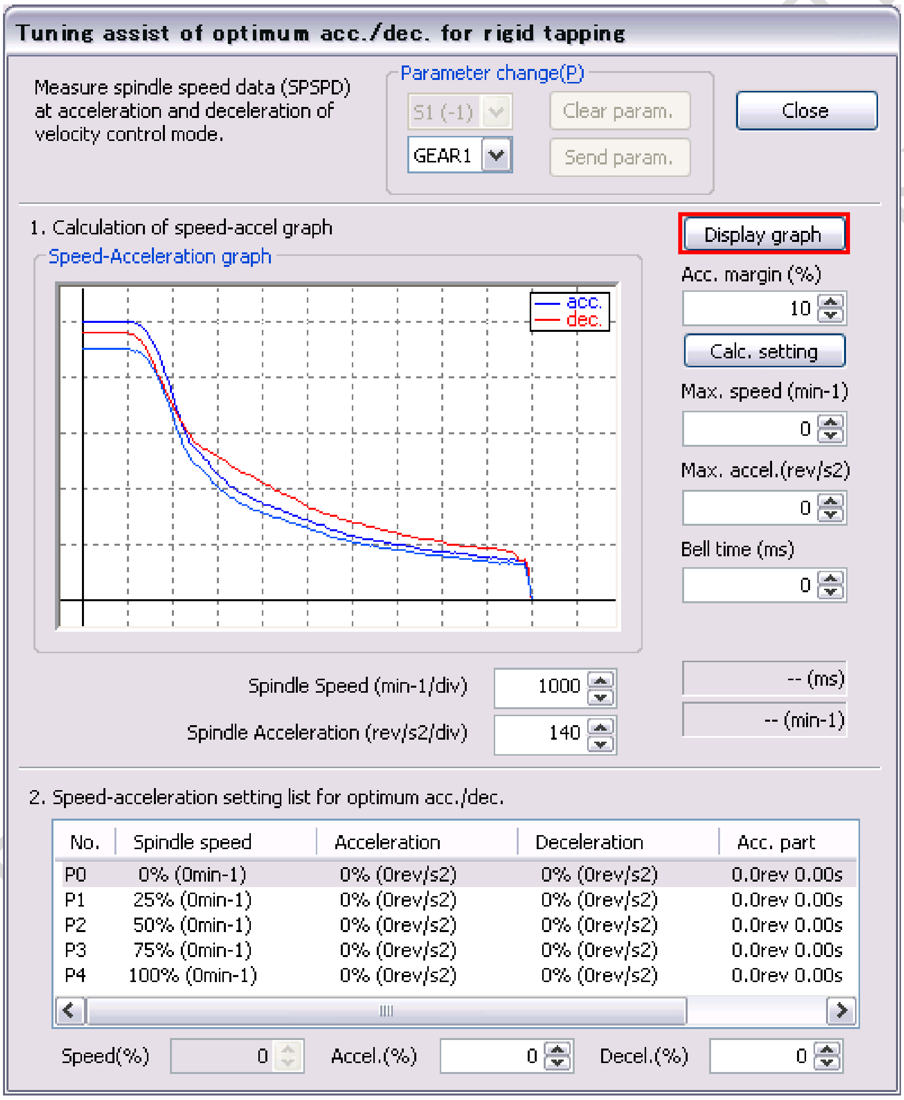
An example measurement program (maximum spindle speed is $10000min^{-1}$)

M03 S0;	Input of motor excitation command
G04 X2.0;	Time for excitation rising
S10000;	Acceleration command until $10000min^{-1}$
G04 X1.5;	Time for acceleration
S0;	Deceleration command until $0min^{-1}$
G04 X1.5;	Time for deceleration
M05;	Stop of motor excitation command
M30;	Program end



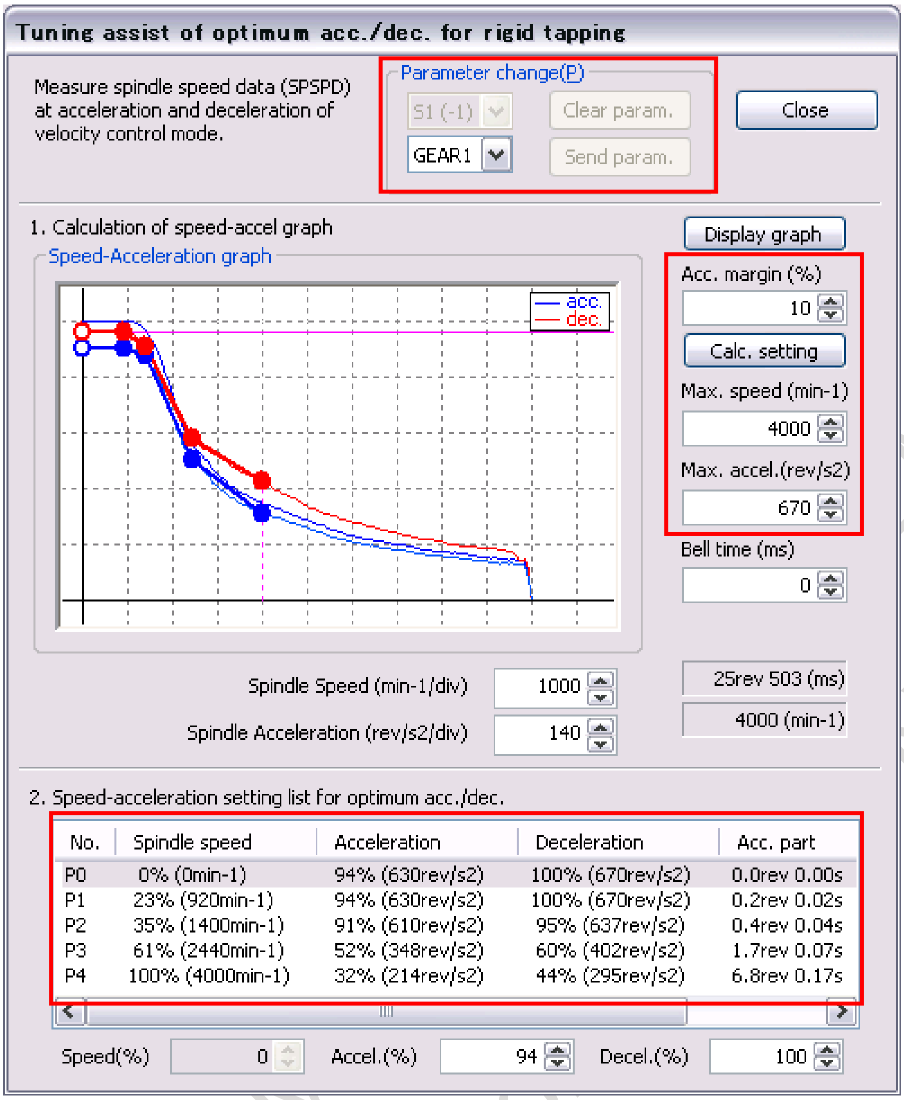
(2)Setup of “Tuning assist of rigid tapping acc./dec.”

Select “Rigid tapping tuning” from “Tool” menu of measurement spindle speed waveform, and select “Optimum acc./dec. for rigid tapping”.



Push “Display graph” button on tuning assist window, then speed-acceleration characteristics calculated by measured spindle waveform are displayed.

(3)Parameter setting of optimum acc./dec. for rigid tapping

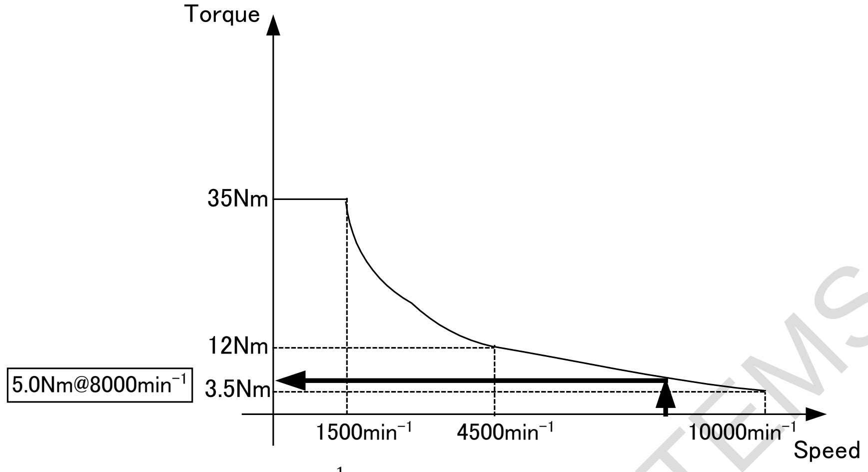


Set “Acc. margin” in consideration of the load of rigid tapping. (Standard setting: 5 to 10%)
When “Calc. setting” button is pushed after the value is input to “Max. speed”, the setting values are calculated according to speed-acceleration curve.
After the initial values are calculated, select the used gear setting in group box of “Parameter change”. Then, “function bit” and “Max. acc.” and “allowable acc.” and “allowable dec.” are input to target parameters of CNC by pushing “Send param.” button.

After the above-mentioned setting is over, check the operation of rigid tapping.

Example of the adjustment (Manual tuning)

Assuming that we use the spindle motor, which has following character, at 8000min^{-1} for rigid tapping.



This motor has small torque at 8000min^{-1}. When the acceleration is constant value in all speed, the torque in low speed area is not utilized enough, because the tolerable acceleration is limited by the torque in high speed area.

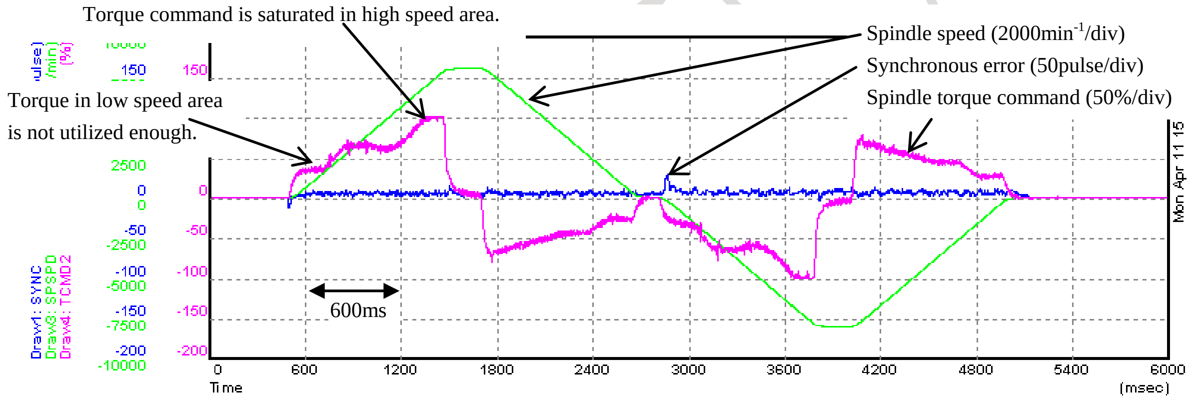


Fig. 7.2.8 (f) Example of measurement result without optimum acc./dec. for rigid tapping

The adjustment of optimum acc./dec. for rigid tapping is performed by following processes.

(1) The initial setting of the acceleration/deceleration pattern

(2) Tune the parameters so that the spindle torque command reaches near 100% in all speed area.

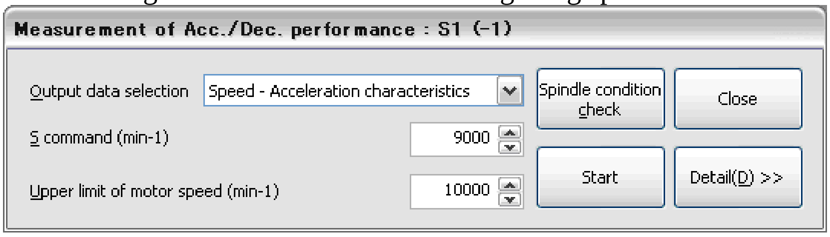
The detail of these processes is explained below.

> **NOTE**
> 1 Perform the adjustment of optimum acc./dec. for rigid tapping after the adjustment of position gain and velocity loop gain. (Refer to the “Adjustment Procedure” of "Rigid Tapping" in “FANUC AC SPINDLE PARAMETER MANUAL (B-65280EN)“ for the details of the adjustment of the gain.)
> 2 In this document, the example treats gear 1. The same procedure can be performed for other gear number.

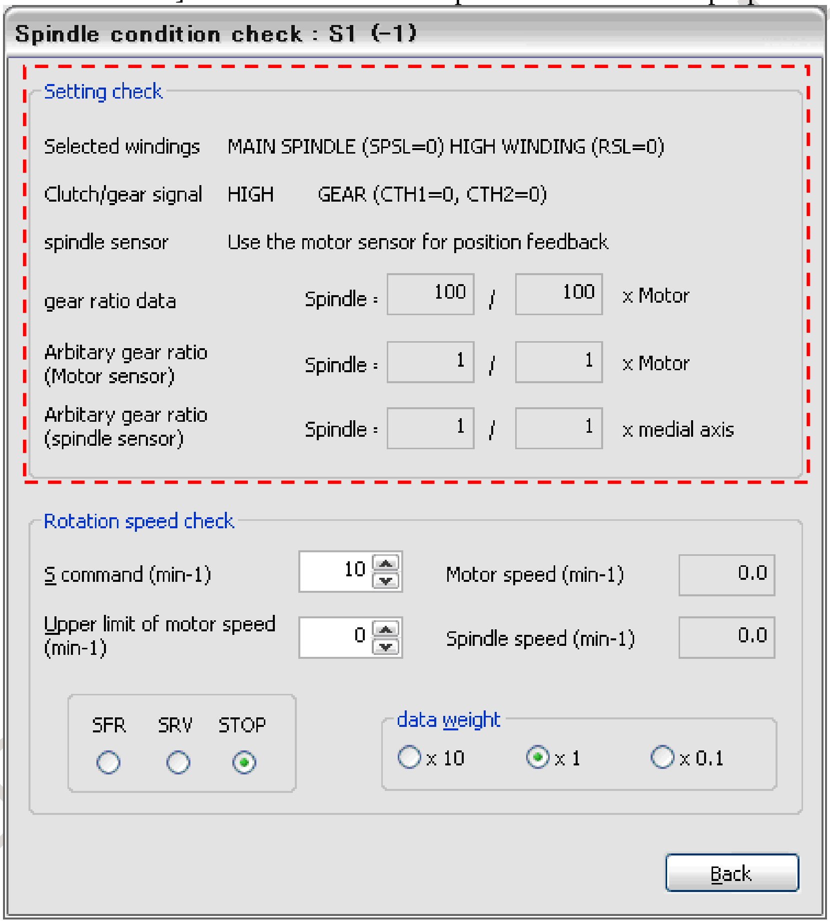
(1)The initial setting of the acceleration/deceleration pattern

Please use “Acc./Dec performance measurement” for spindle tuning in SERVO GUIDE to determine the initial settings for optimum acc./dec. for rigid tapping.

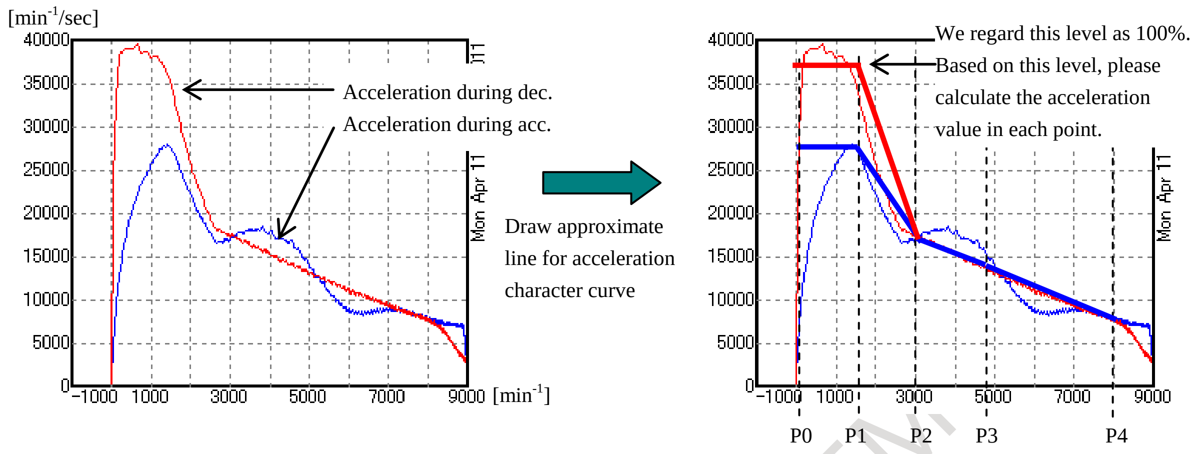
Please open Graph Window and select [Tool] -> [Spindle Tuning] -> [Acc./Dec performance measurement]. The following window will be shown after getting spindle information.



Click [Spindle condition check] and confirm whether spindle conditions are proper or not.



Go back to the measurement window and after setting S command and upper limit of motor speed, click "Start". Please set higher value than the maximum rigid tapping speed in S command box, if you can.



Please estimate the initial setting values from the measurement result. Please draw approximate line through each setting point (P0-P4) to determine the acceleration character during acc./dec.. The above data shows the character in acc./dec. with maximum torque. So available torque in rigid tapping will be less than this character. Please set the initial value for optimum acc./dec. for rigid tapping as 70% or so of approximate line.

Example)The maximum acceleration (parameter No.11421) can be seen at 0-1500min^{-1} in deceleration. It's value is

$$37800(0 \sim 1500\,\mathrm{min}^{-1})[\mathrm{min}^{-1}/\mathrm{sec}] \div 60[\mathrm{sec}] \times 0.7 = 441[rev/\mathrm{sec}^2]$$

The rest of acceleration settings should be set as the proportion of this value.

Example)Parameter No.11441 is calculated from acceleration value (27600min^{-1}/sec) at 0-1500min^{-1} in acceleration as follows:

$$27600[\mathrm{min}^{-1}/\mathrm{sec}]/37800[\mathrm{min}^{-1}/\mathrm{sec}] \times 100 = 73[\%]$$

Table 7.2.8 (c) Initial settings for optimum acc./dec. for rigid tapping

Gear	Maximum acceleration [rev/sec^2]	Maximum spindle speed [min^{-1}]	Acc. setting point	Spindle speed (Ratio to max. speed [%])	Acc. setting (Ration to max. acc. [%]) Forward / Reverse	
					During acc.	During dec.
Gear 1	No.11421=441	No.5241=8000	P0	None(0%)	No.11441=73	No.11461=100
			P1	No.11429=19	No.11442=73	No.11462=100
			P2	No.11430=38	No.11443=46	No.11463=46
			P3	No.11431=60	No.11444=36	No.11464=36
			P4	None(100%)	No.11445=20	No.11465=20

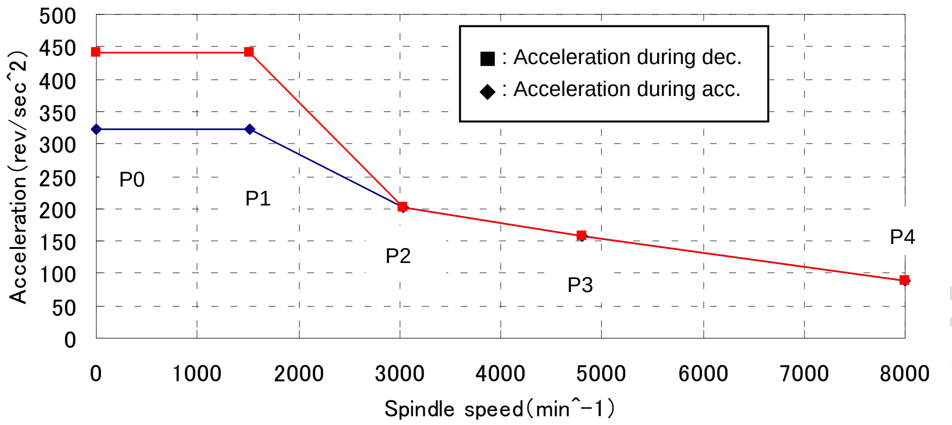


Fig. 7.2.8 (g) Initial settings for optimum acc./dec. for rigid tapping

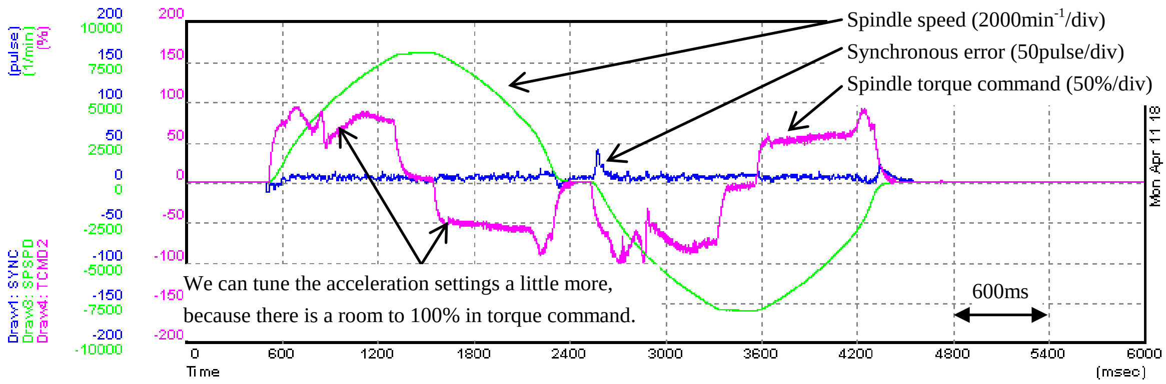


Fig. 7.2.8 (h) Example of measurement result after initial setting of optimum acc./dec. for rigid tapping

(2) Tune the parameters so that the spindle torque command reaches near 100% in all speed area

Capture the data during the rigid tapping under initial setting of optimum acc./dec. for rigid tapping by using SERVO GUIDE. The data of spindle speed (SPSPD) and torque command (TCMD2(NOTE)) should be captured. After confirming the measurement result with initial setting, please tune the acceleration parameters so that the spindle torque command reaches near 100% in all speed area. When you tune them, please use the test program, which has the assumed maximum rigid tapping speed, and start tuning from the high speed point (P4) first, and continue tuning the low speed points in order.

NOTE

TCMD2 shows the torque command rate, in which the maximum torque, considering torque limit value at each speed, is 100%.
To capture this data, following SERVO GUIDE and spindle software is required.
SERVO GUIDE: Ver.8.10 or later
Spindle soft: 9DA0 series 14 edition or later
If you can't use TCMD2, please use TCMD. However, TCMD does not consider torque limit value. Thus, TCMD might be saturated at a value which is lower than 100%. You can see whether torque command is saturated or not by watching the data of bit12 of "SFLG1". Please watch them together, if you use TCMD.

Table 7.2.8 (d) After tuning settings for optimum acc./dec. for rigid tapping

Gear	Maximum acceleration [rev/sec^2]	Maximum spindle speed [min^{-1}]	Acc. setting point	Spindle speed (Ratio to max. speed [%])	Acc. setting (Ration to max. acc. [%]) Forward / Reverse	
					During acc.	During dec.
Gear 1	No.11421=441	No.5241=8000	P0	None(0%)	No.11442=73	No.11462=100
			P1	No.11429=19	No.11442=73	No.11462=100
			P2	No.11430=38	No.11443=52	No.11463=60
			P3	No.11431=60	No.11444=47	No.11464=55
			P4	None(100%)	No.11445=20	No.11465=40

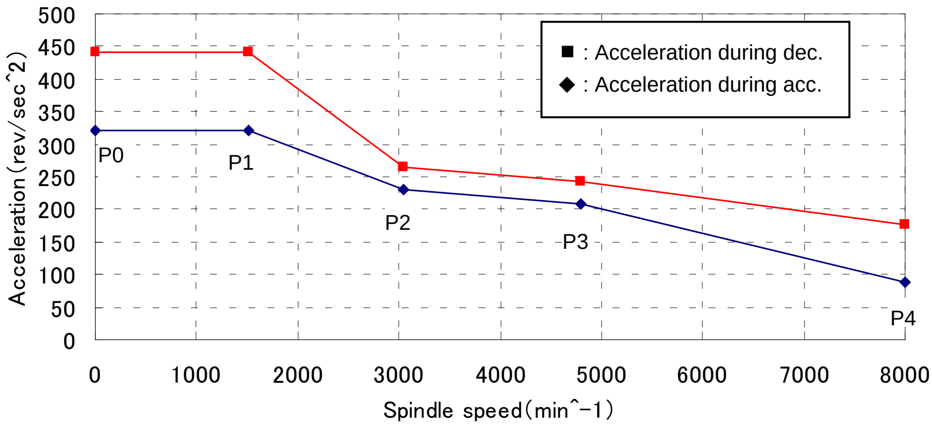


Fig. 7.2.8 (i) After tuning settings for optimum acc./dec. for rigid tapping

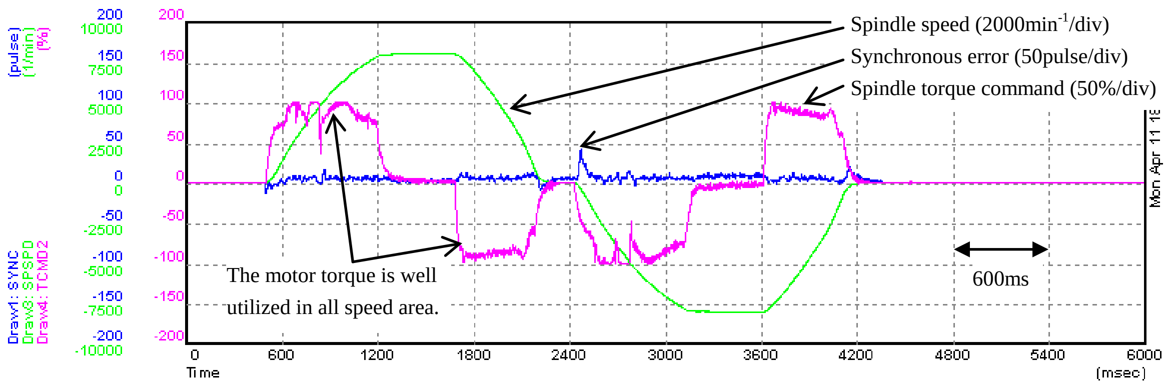


Fig. 7.2.8 (j) Example of measurement result after tuning of optimum acc./dec. for rigid tapping

In the above example, the acceleration time and deceleration time are reduced as follows by applying optimum acc./dec. for rigid tapping.

	Function OFF	Function ON
Acceleration time	976 ms	720 ms
Deceleration time	976 ms	588 ms

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11420								RAU

[Input type] Parameter input
[Data type] Bit path

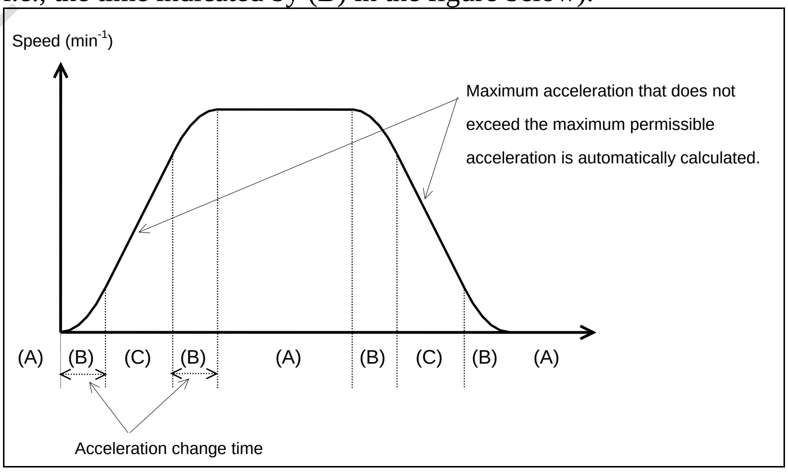
#0 RAU Optimum torque acceleration/deceleration function for rigid tapping is
0: Disabled.
1: Enabled.

11421	**Maximum acceleration of the optimum acceleration/deceleration for rigid tapping (gear 1)**
11422	**Maximum acceleration of the optimum acceleration/deceleration for rigid tapping (gear 2)**
11423	**Maximum acceleration of the optimum acceleration/deceleration for rigid tapping (gear 3)**
11424	**Maximum acceleration of the optimum acceleration/deceleration for rigid tapping (gear 4)**

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] rev/sec^2
[Valid data range] 0 to 10000.0
These parameters set maximum accelerations.

11425	**Acceleration change time of bell-shaped acceleration/deceleration in optimum acceleration/deceleration for rigid tapping (gear 1)**
11426	**Acceleration change time of bell-shaped acceleration/deceleration in optimum acceleration/deceleration for rigid tapping (gear 2)**
11427	**Acceleration change time of bell-shaped acceleration/deceleration in optimum acceleration/deceleration for rigid tapping (gear 3)**
11428	**Acceleration change time of bell-shaped acceleration/deceleration in optimum acceleration/deceleration for rigid tapping (gear 4)**

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] msec
[Valid data range] 0 to 200
These parameters set the acceleration change time of bell-shaped acceleration/deceleration in optimum acceleration/deceleration for rigid tapping (time taken for the change from the constant speed state (A) to the acceleration state (C) with the acceleration calculated from the optimum acceleration/deceleration for rigid tapping, i.e., the time indicated by (B) in the figure below).



11429	**Spindle speed at P1 in optimum acceleration/deceleration for rigid taping (gear 1)**

11430	**Spindle speed at P2 in optimum acceleration/deceleration for rigid taping (gear 1)**

11431	**Spindle speed at P3 in optimum acceleration/deceleration for rigid taping (gear 1)**

11432	**Spindle speed at P1 in optimum acceleration/deceleration for rigid taping (gear 2)**

11433	**Spindle speed at P2 in optimum acceleration/deceleration for rigid taping (gear 2)**

11434	**Spindle speed at P3 in optimum acceleration/deceleration for rigid taping (gear 2)**

11435	**Spindle speed at P1 in optimum acceleration/deceleration for rigid taping (gear 3)**

11436	**Spindle speed at P2 in optimum acceleration/deceleration for rigid taping (gear 3)**

11437	**Spindle speed at P3 in optimum acceleration/deceleration for rigid taping (gear 3)**

11438	**Spindle speed at P1 in optimum acceleration/deceleration for rigid taping (gear 4)**

11439	**Spindle speed at P2 in optimum acceleration/deceleration for rigid taping (gear 4)**

11440	**Spindle speed at P3 in optimum acceleration/deceleration for rigid taping (gear 4)**

[Input type] Parameter input
[Data type] Byte spindle
[Unit of data] %
[Valid data range] 0 to 100

These parameters set the spindle speeds at P1 to P3 of acceleration points P0 to P4 as ratios to the maximum spindle speed (parameters Nos. 5241 to 5244). The spindle speed at P0 is 0, while the spindle speed at P4 is the maximum spindle speed. Any acceleration setting points where 0 is set will be skipped.

11441	**Permissible acceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 1)**

11442	**Permissible acceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 1)**

11443	**Permissible acceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 1)**

11444	**Permissible acceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 1)**

11445	**Permissible acceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 1)**

11446	**Permissible acceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 2)**

11447	**Permissible acceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 2)**

11448	**Permissible acceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 2)**

11449	**Permissible acceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 2)**

11450	**Permissible acceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 2)**

11451	Permissible acceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 3)
11452	Permissible acceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 3)
11453	Permissible acceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 3)
11454	Permissible acceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 3)
11455	Permissible acceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 3)
11456	Permissible acceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 4)
11457	Permissible acceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 4)
11458	Permissible acceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 4)
11459	Permissible acceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 4)
11460	Permissible acceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 4)

[Input type] Parameter input
[Data type] Byte spindle
[Unit of data] %
[Valid data range] 0 to 100

These parameters set the permissible accelerations at acceleration setting points P0 to P4 as ratios to the maximum acceleration (parameters Nos. 11421 to 11424). At any acceleration setting points where 0 is set, 100% is assumed.

11461	Permissible deceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 1)
11462	Permissible deceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 1)
11463	Permissible deceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 1)
11464	Permissible deceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 1)
11465	Permissible deceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 1)
11466	Permissible deceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 2)
11467	Permissible deceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 2)
11468	Permissible deceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 2)
11469	Permissible deceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 2)
11470	Permissible deceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 2)
11471	Permissible deceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 3)

11472	Permissible deceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 3)

11473	Permissible deceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 3)

11474	Permissible deceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 3)

11475	Permissible deceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 3)

11476	Permissible deceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 4)

11477	Permissible deceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 4)

11478	Permissible deceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 4)

11479	Permissible deceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 4)

11480	Permissible deceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 4)

[Input type] Parameter input
[Data type] Byte spindle
[Unit of data] %
[Valid data range] 0 to 100

These parameters set the permissible decelerations at acceleration setting points P0 to P4 as ratios to the maximum acceleration (parameters Nos. 11421 to 11424). At any acceleration setting points where 0 is set, 100% is assumed.

Alarm and message

Number	Message	Description
DS1711	ILLEGAL ACC. PARAMETER (RIGID TAPPING OPTIMUM ACC/DEC)	The allowable acceleration parameter of optimum acceleration/deceleration for rigid tapping is incorrectly set. The probable causes are shown below. <1> The ratio of deceleration to acceleration is less than one-third. <2> The time required for deceleration to a speed of 0 exceeds the maximum value. <3> The maximum acceleration (parameters Nos. 11421 to 11424) is 0.

Reference item

Manual name	Item name
OPERATOR'S MANUAL	
(B-64694EN-1)	Rigid tapping
(B-64694EN-2)	Rigid tapping

7.2.9 Rapid Traverse Acceleration/deceleration before Interpolation

Overview

In linear type rapid traverse, the positioning path is linear. If look-ahead acceleration/deceleration before interpolation is enabled, however, acceleration/deceleration can be performed with the linear path kept.

When the following conditions are met, look-ahead acceleration/deceleration before interpolation is enabled for rapid traverse.

- When rapid traverse is of linear interpolation type (Bit 1 (LRP) of parameter No. 1401 is set to 1)
- When look-ahead acceleration/deceleration before interpolation is enabled for rapid traverse (Bit 5 (FRP) of parameter No. 19501 is set to 1)
- A value other than 0 is set in parameter No. 1671 for an axis.
- The AI contour control mode is set.

Deceleration and stop is always performed between blocks for which look-ahead acceleration/deceleration before interpolation is enabled.
Rapid traverse block overwrapping is disabled.

- Bell-shaped acceleration/deceleration before interpolation for rapid traverse

If parameter No. 1672 is set to bell-shaped change time when look-ahead acceleration/deceleration before interpolation is enabled for rapid traverse, acceleration/deceleration becomes bell-shaped.

- Enabling acceleration/deceleration after interpolation for acceleration/deceleration before interpolation

If parameter No. 11242 is set for each axis to the time constant of acceleration/deceleration after interpolation for rapid traverse look-ahead acceleration/deceleration before interpolation, it is possible to enable acceleration/deceleration after interpolation is for look-ahead acceleration/deceleration before interpolation.

> **NOTE**
> When enabling acceleration/deceleration after interpolation for rapid traverse acceleration/deceleration before interpolation, be sure to set the same time constant for all axes except a special purpose. Otherwise, the correct line cannot be obtained.

- Acceleration/deceleration before interpolation for rapid traverse in AICC mode off

If all of the following conditions are satisfied, look-ahead acceleration/deceleration before interpolation for rapid traverse is enabled in AICC mode off.

- AICC I or AICC II+ options is valid.
- The parameters for which look-ahead acceleration/deceleration before interpolation for rapid traverse is enabled are set.
- The setting of look-ahead acceleration/deceleration before interpolation for rapid traverse in the AICC mode off is enabled (Bit 4 (ARB) of parameter No.11240 is set to 1.)

> **NOTE**
> In the function which look-ahead acceleration/deceleration before interpolation is invalid, this function is also disabled.

- Acceleration/deceleration before interpolation for rapid traverse in rigid tapping.

When look-ahead acceleration/deceleration before interpolation for rapid traverse is valid, if the setting of look-ahead acceleration/deceleration before interpolation for rapid traverse in rigid tapping is enabled (Bit 5 (RRB) of parameter No.11240 is set to 1), look-ahead acceleration/deceleration before interpolation for rapid traverse in rigid tapping is valid. (Rapid traverse in rigid tapping is the motion of dotted line in Fig.7.2.9 .)

However, the specification of acceleration/deceleration in the following functions follows the specification of each function.

- Optimum Acceleration/Deceleration for Rigid Tapping
- Rigid Tapping with Servo Motor

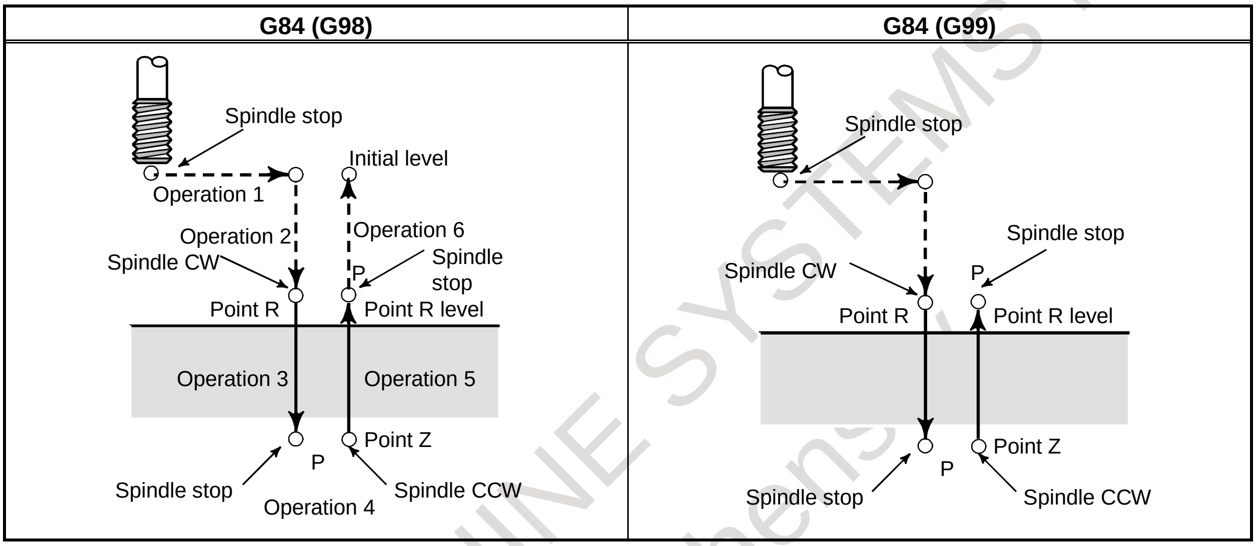


Fig.7.2.9 Rigid tapping command

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401							LRP	

[Input type] Parameter input
[Data type] Bit path

#1 LRP Positioning (G00)
0: Positioning is performed with non-linear type positioning so that the tool moves along each axis independently at rapid traverse.
1: Positioning is performed with linear interpolation so that the tool moves in a straight line.
When using 3-dimensional coordinate system conversion, set this parameter to 1.

1671	Maximum allowable acceleration rate in acceleration/deceleration before interpolation for linear rapid traverse for each axis, or maximum allowable reference acceleration rate in optimum torque acceleration/deceleration

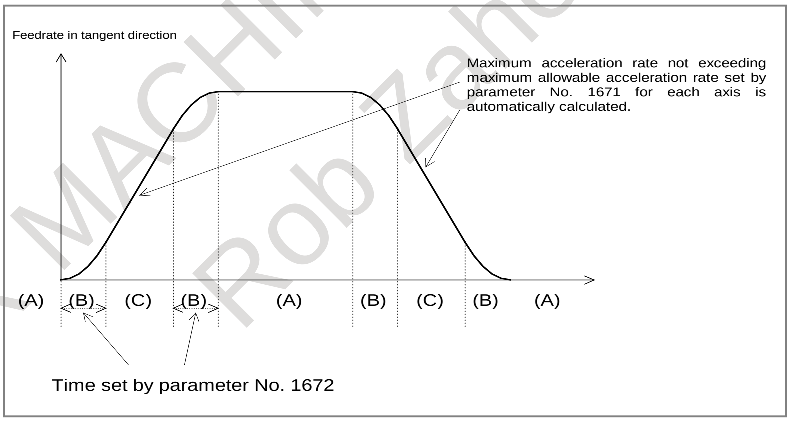
[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

(1) Set a maximum allowable acceleration rate in look-ahead acceleration/ deceleration before interpolation for linear rapid traverse.
If a value greater than 100000.0, the value is clamped to 100000.0.
If 0 is set, the specification of the following is assumed:
1000.0 mm/sec^2
100.0 inch/sec^2
100.0 degrees/sec^2
If 0 is specified for all axes, however, look-ahead acceleration/deceleration before interpolation is not performed.
(2) Maximum allowable reference acceleration rate in optimum torque acceleration/deceleration

1672	Acceleration change time of bell-shaped acceleration/deceleration before interpolation for linear rapid traverse, or acceleration change time of bell-shaped acceleration/deceleration in optimum torque acceleration/deceleration

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 200

(1) Set an acceleration change time of bell-shaped acceleration/ deceleration for linear rapid traverse (time for changing from the state of constant feedrate (A) to the state of constant acceleration/deceleration (C) at the acceleration rate calculated from the acceleration rate set in parameter No. 1671: time of (B) in the figure below).
(2) Set an acceleration change time of bell-shaped acceleration/ deceleration in optimum torque acceleration/deceleration (time for changing from the state of constant feedrate (A) to the state of acceleration/deceleration (C) at the acceleration rate calculated from optimum torque acceleration/deceleration: time of (B) in the figure).



	#7	#6	#5	#4	#3	#2	#1	#0
11240			RRB	ARB				

[Input type] Parameter input
[Data type] Bit path

#4 ARB Look-ahead acceleration/deceleration before interpolation for rapid traverse in AICC mode off is:
0: Disabled.
1: Enabled.

NOTE
This parameter is enabled when the parameters for which look-ahead acceleration/deceleration before interpolation for rapid traverse is enabled are set. (Refer to bit 5 (FRP) of parameter No.19501.)

#5 RRB Look-ahead acceleration/deceleration before interpolation for rapid traverse in rigid tapping is:
0: Disabled.
1: Enabled.

NOTE
This parameter is enabled when look-ahead acceleration/deceleration before interpolation for rapid traverse is valid. (Refer to bit 5 (FRP) of parameter No.19501.)

11242	Time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

This parameter is used for the time constant of look-ahead acceleration/deceleration before interpolation in rapid traverse.
Be sure to specify the same time constant value for all axes except for a special application. If different time constants are set, a correct linear line cannot be obtained.

	#7	#6	#5	#4	#3	#2	#1	#0
19501			FRP					

[Input type] Parameter input
[Data type] Bit path

#5 FRP Linear rapid traverse is:
0: Acceleration/deceleration after interpolation
1: Look-ahead acceleration/deceleration before interpolation

Set a maximum allowable acceleration rate for each axis in parameter No. 1671.
When using look-ahead bell-shaped acceleration/deceleration before interpolation, set an acceleration rate change time in parameter No. 1672.

When this parameter is set to 1, look-ahead acceleration/deceleration before interpolation is also applied to rapid traverse if all conditions below are satisfied. At this time, acceleration/deceleration after interpolation is not applied.
- Bit 1 (LRP) of parameter No. 1401 is set to 1: Linear interpolation type positioning
- A value other than 0 is set in parameter No. 1671 for an axis.
- The AI contour control mode is set.

If all of these conditions are not satisfied, acceleration/deceleration after interpolation is applied.

7.2.10 Time constant of acceleration / deceleration after interpolation switching function by signal

Overview

Time constant of acceleration / deceleration after interpolation can be switched by input signal of PMC.

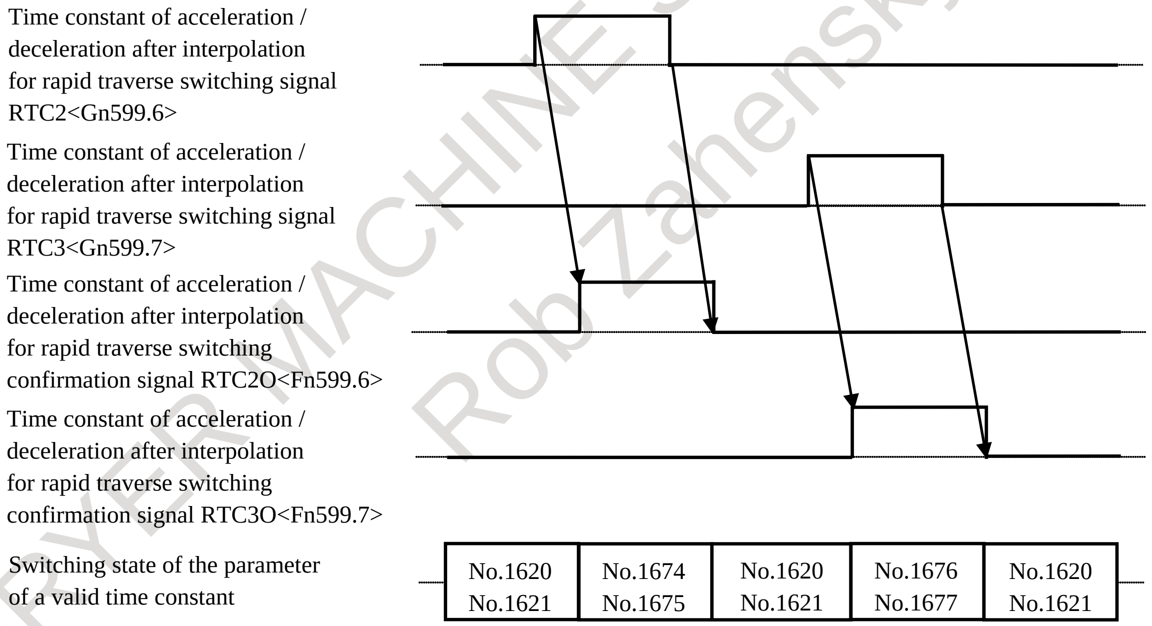
Explanation

In this function, time constant of acceleration / deceleration after interpolation for rapid traverse and cutting feed can be switched by input signal of PMC.

- Time constant of acceleration / deceleration after interpolation for rapid traverse switching

A time constant of acceleration / deceleration after interpolation for rapid traverse can be switched to either of the three following kinds by time constant of acceleration / deceleration after interpolation for rapid traverse switching signals RTC2, RTC3<Gn599.6, Gn599.7>.
- The usual time constant for rapid traverse (Parameter Nos. 1620 and 1621)
- The second time constant for rapid traverse (Parameter Nos. 1674 and 1675)
- The third time constant for rapid traverse (Parameter Nos. 1676 and 1677)

Example of time chart for time constant of acceleration / deceleration after interpolation for rapid traverse switching

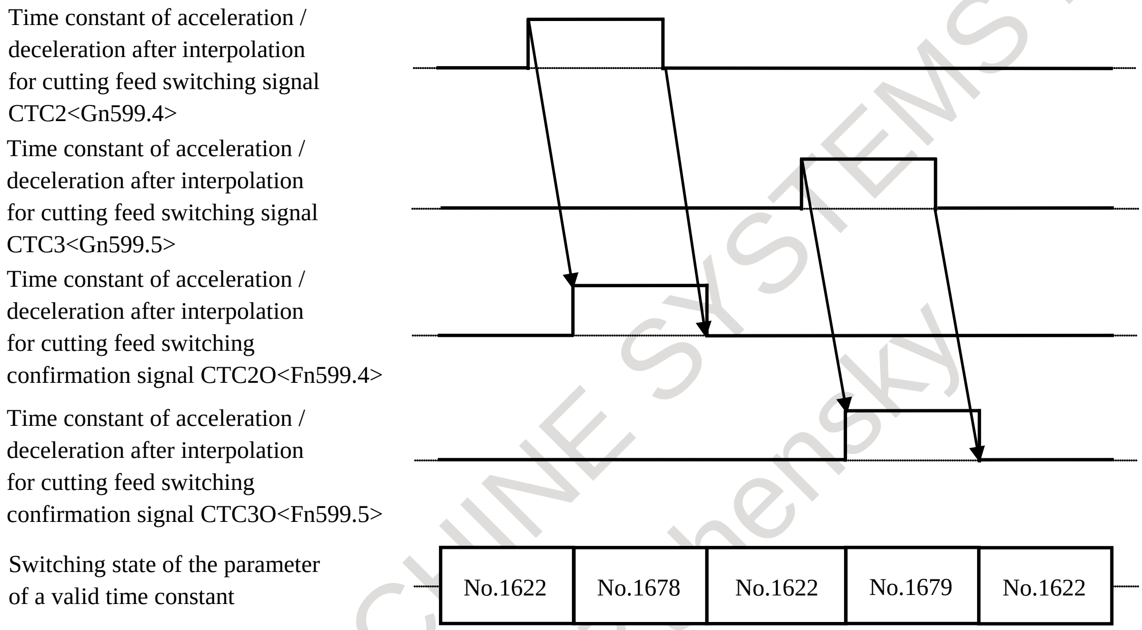


Confirm an effective parameter by time constant of acceleration / deceleration after interpolation for rapid traverse switching confirmation signal RTC2O, RTC3O.

- Time constant of acceleration / deceleration after interpolation for cutting feed switching

A time constant of acceleration / deceleration after interpolation for cutting feed can be switched to either of the three following kinds by time constant of acceleration / deceleration after interpolation for cutting feed switching signals CTC2, CTC3<Gn599.4, Gn599.5>.
- The usual time constant for cutting feed (Parameter No. 1622)
- The second time constant for cutting feed (Parameter No. 1678)
- The third time constant for cutting feed (Parameter No. 1679)

Example of time chart for time constant of acceleration / deceleration after interpolation for cutting feed switching



Confirm an effective parameter by time constant of acceleration / deceleration after interpolation for cutting feed switching confirmation signal CTC2O, CTC3O.

NOTE

1) If -1 is set to parameters Nos. 1674 or 1676 or a value not within the valid data range is specified in these parameters, the second or the third time constant of acceleration / deceleration after interpolation for rapid traverse is invalid in its axis.
2) If -1 is set to parameters Nos. 1678 or 1679 or a value not within the valid data range is specified in these parameters, the second or the third time constant of acceleration / deceleration after interpolation for cutting feed is invalid in its axis.
3) To switch a time constant of acceleration / deceleration after interpolation, all the valid axes should stop. When the axis has not stopped, the time constant is switched at the next program block after stopping all the valid axes.
4) When an axis has been moved to other path by flexible path axis assignment function, a time constant of acceleration / deceleration after interpolation is switched by RTC2, RTC3, CTC2 and CTC3 of the path after moving.
5) A slave axis of synchronous control is used the same kind of acceleration / deceleration after interpolation as a master axis. However, as for the time constant, each one is used. When the master axis is a valid axis for this function, and the slave axis is an invalid axis, as for the time constant of the slave axis, the usual time constant is used.

6) A movement axis of composite control is used the same kind of acceleration / deceleration after interpolation as a specified axis. However, as for the time constant, each one is used. When the specified axis is a valid axis for this function, and the movement axis is an invalid axis, as for the time constant of the movement axis, the usual time constant is used.

7) In the following cases, a time constant for this function is invalid.
   - A time constant of acceleration / deceleration after interpolation for manual handle retrace is valid

8) In the following cases, a time constant for rapid traverse for this function is invalid.
   - As for the axes (a master axis and a slave axis) in superimposed control (with speed control)
   - A time constant of acceleration / deceleration after interpolation for positioning by optimum accelerations is valid
   - Acceleration / deceleration before interpolation for rapid traverse of AI contour control is valid

9) In the following cases, a time constant for cutting feed for this function is invalid.
   - A time constant of acceleration / deceleration after interpolation for the skip is valid
   - A time constant of acceleration / deceleration after interpolation for rigid tapping is valid
   - A time constant of acceleration / deceleration after interpolation for spindle control with servo motor is valid
   - A time constant of exponential acceleration / deceleration after interpolation in cutting feed for PMC axis control is valid
   - Acceleration / deceleration before interpolation of AI contour control is valid

7.2.10.1 Time constant of acceleration/deceleration after interpolation in the mode of acceleration/deceleration before interpolation switching function by PMC signal

Outline

A time constant of acceleration/deceleration after interpolation in the mode of acceleration/deceleration before interpolation can be switched by PMC signal.

Explanation

In this function, the following time constant can be switched by PMC signal.

- Time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse
- Time constant of acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode

- Time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse switching

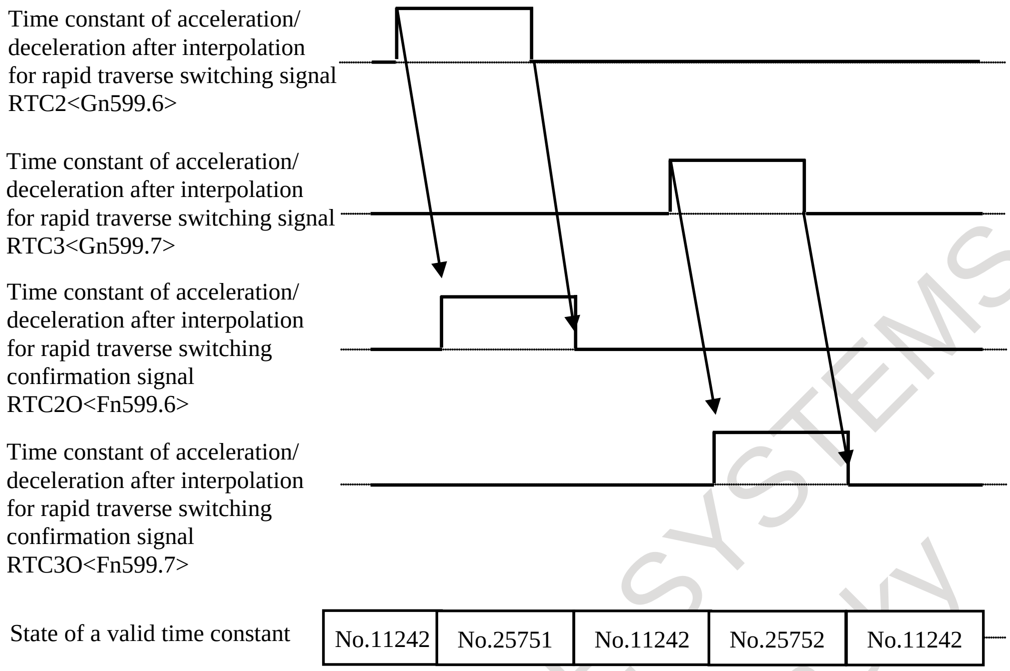
When parameter BRT(No.25750#0) is set to 1, a time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse can be switched to either of the following three parameters by time constant of acceleration/deceleration after interpolation for rapid traverse switching signals RTC2, RTC3<Gn599.6, Gn599.7>.

Set parameter No. 11242 to a value from 1 to 4000, when executing “Time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse switching”.

- The time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse
(Parameter No. 11242)
- The second time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse
(Parameter No. 25751)

- The third time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse
(Parameter No. 25752)

Example)

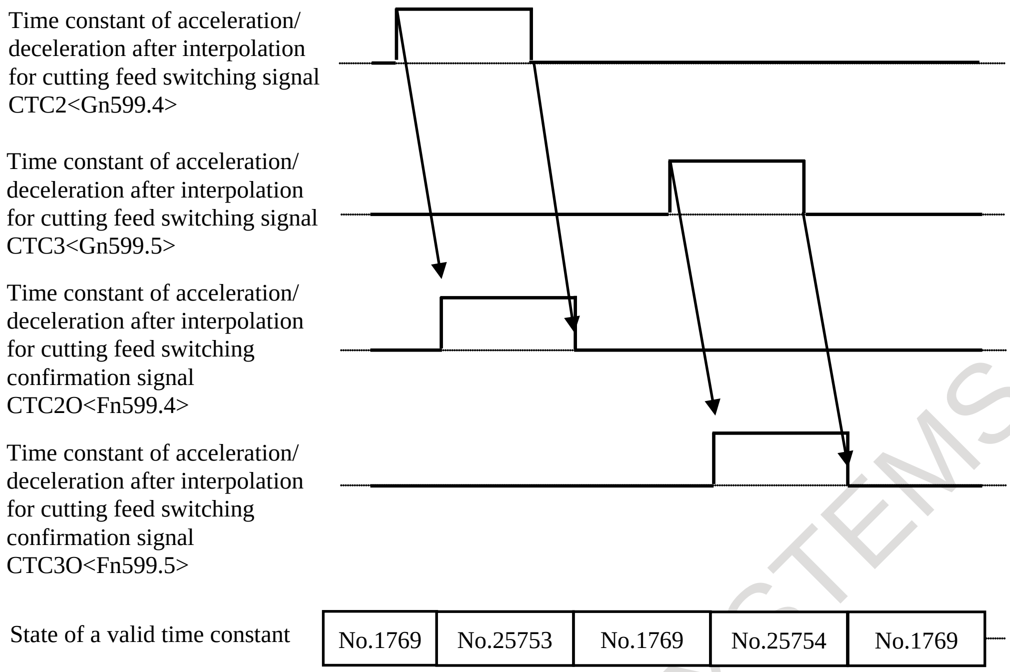


- Time constant of acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode switching

When parameter BCT(No.25750#1) is set to 1, a time constant of acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode can be switched to either of the following three parameters by time constant of acceleration/deceleration after interpolation for cutting feed switching signals CTC2, CTC3<Gn599.4, Gn599.5>.

- The time constant of acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode
(Parameter No. 1769)
- The second time constant of acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode
(Parameter No. 25753)
- The third time constant of acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode
(Parameter No. 25754)

Example)



NOTE

1) In a function of not supporting time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation, this function is disabled.
2) When the parameter No.25751 or 25752 is set to -1, the second or third time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse is disabled.
3) When the parameter No.25753 or 25754 is set to -1, the second or third time constant of acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode is disabled.
4) To switch a time constant of acceleration/deceleration after interpolation, all the valid axes must be stopped. When the axis has not stopped, the time constant is switched at the next program block after stopping all the valid axes.
5) When the parameters Nos.25751 to 25754 are changed during switching to a time constant of acceleration/deceleration after interpolation for this function, the time constant is also changed.
6) When an axis has been moved to other path by flexible path axis assignment function, a time constant of acceleration/deceleration after interpolation is switched by RTC2, RTC3, CTC2 and CTC3 of the path after moving.
7) A slave axis of the synchronous control is used the same kind of acceleration/deceleration after interpolation as a master axis. However, as for the time constant, each one is used. When the master axis is a valid axis for this function, and the slave axis is an invalid axis, as for the time constant of the slave axis, the usual time constant is used.
8) A movement axis of the composite control is used the same kind of acceleration/deceleration after interpolation as a specified axis. However, as for the time constant, each one is used. When the specified axis is a valid axis for this function, and the movement axis is an invalid axis, as for the time constant of the movement axis, the usual time constant is used.
9) In the following case, a time constant for this function is valid.
   - A time constant of acceleration/deceleration after interpolation in the acceleration/deceleration before interpolation mode for the canned cycle for drilling is valid
   - A time constant of acceleration/deceleration after interpolation in the acceleration/deceleration before interpolation mode for the automatic corner speed control is valid

10) In the following case, a time constant for this function is invalid.
   - A time constant of acceleration/deceleration after interpolation for the manual handle retrace is valid
11) In the following cases, a time constant for rapid traverse for this function is invalid.
   - As for the axes (a master axis and a slave axis) in the superimposed control (with speed control)
12) In the following cases, a time constant for cutting feed for this function is invalid.
   - A time constant of acceleration/deceleration after interpolation for the skip function is valid
   - A time constant of acceleration/deceleration after interpolation for the rigid tapping is valid

Signal

Time constant of acceleration / deceleration after interpolation for rapid traverse switching signals RTC2, RTC3 <Gn599.6, Gn599.7>

[Classification] Input signal

[Function] It changes to selected the kind of a time constant of acceleration / deceleration after interpolation for rapid traverse.

[Operation] The time constant of acceleration/deceleration after interpolation for rapid traverse is switched as the following table.

Table 7.2.10 (e) Switching the time constant of acc/dec after interpolation for rapid traverse

RTC2 <Gn599.6>	RTC3 <Gn599.7>	BRT (No.25750#0)	time constant of acc/dec after interpolation for rapid traverse	time constant of acc/dec after interpolation of acc/dec before interpolation in rapid traverse
0	0	0	No.1620, No.1621	No.11242
1	0	0	No.1674, No.1675	No.11242
0	1	0	No.1676, No.1677	No.11242
1	1	0	No.1674, No.1675	No.11242
0	0	1	No.1620, No.1621	No.11242
1	0	1	No.1674, No.1675	No.25751
0	1	1	No.1676, No.1677	No.25752
1	1	1	No.1674, No.1675	No.25751

Time constant of acceleration / deceleration after interpolation for cutting feed switching signals CTC2, CTC3 <Gn599.4, Gn599.5>

[Classification] Input signal

[Function] It changes to selected the kind of a time constant of acceleration / deceleration after interpolation for cutting feed.

[Operation] Time constant of acceleration/deceleration after interpolation for cutting feed is switched as the following table.

Table 7.2.10 (f) Switching the time constant of acc/dec after interpolation for cutting feed

CTC2 <Gn599.4>	CTC3 <Gn599.5>	BCT (No.25750#1)	time constant of acc/dec after interpolation for cutting feed	time constant of acc/dec after cutting feed interpolation in the acc/dec before interpolation mode
0	0	0	No.1622	No.1769
1	0	0	No.1678	No.1769
0	1	0	No.1679	No.1769
1	1	0	No.1678	No.1769
0	0	1	No.1622	No.1769
1	0	1	No.1678	No.25753
0	1	1	No.1679	No.25754
1	1	1	No.1678	No.25753

Time constant of acceleration / deceleration after interpolation for rapid traverse switching confirmation signals RTC2O, RTC3O <Fn599.6, Fn599.7>

[Classification] Output signal

[Function] These signals indicate that the change to a time constant of acceleration / deceleration after interpolation for rapid traverse for this function is valid.

[Operation] These signals are set to “1” when time constant for rapid traverse switching is valid.
These signals are set to “0” when time constant for rapid traverse switching is invalid.
RTC2O and RTC3O support as follows.
RTC2O: The second time constant for rapid traverse (Parameter Nos. 1674 and 1675)
RTC3O: The third time constant for rapid traverse (Parameter Nos. 1676 and 1677)

NOTE
When the usual time constant of acceleration / deceleration after interpolation for rapid traverse is not used even if these signals are “1”, a time constant for rapid traverse for this function is invalid.

Time constant of acceleration / deceleration after interpolation for cutting feed switching confirmation signals CTC2O, CTC3O <Fn599.4, Fn599.5>

[Classification] Output signal

[Function] These signals indicate that the change to a time constant of acceleration / deceleration after interpolation for cutting feed for this function is valid.

[Operation] These signals are set to “1” when time constant for cutting feed switching is valid.
These signals are set to “0” when time constant for cutting feed switching is invalid.
CTC2O and CTC3O support as follows.
CTC2O: The second time constant for cutting feed (Parameter No. 1678)
CTC3O: The third time constant for cutting feed (Parameter No. 1679)

NOTE
When the usual time constant of acceleration / deceleration after interpolation for cutting feed is not used even if these signals are “1”, a time constant for cutting feed for this function is invalid.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn599	RTC3	RTC2	CTC3	CTC2				
Fn599	RTC3O	RTC2O	CTC3O	CTC2O				

Parameter

The parameter used by this function is shown as follows.

1674	The second time constant T or T1 of linear acceleration / deceleration or bell-shaped acceleration / deceleration for rapid traverse for each axis

1676	The third time constant T or T1 of linear acceleration / deceleration or bell-shaped acceleration / deceleration for rapid traverse for each axis

[Input type] Parameter input
[Data type] Word axis
[Data unit] msec
[Data range] -1 to 4000

Specify the second or the third time constant of acceleration / deceleration after interpolation for rapid traverse for each axis. As for the axis for which this parameter is set values of -1 or beyond the range, the second or the third time constant for rapid traverse is invalid.

> **NOTE**
> If bit 6 (CTC) of parameter No. 11502 is 1, this parameter is written during moving axes.

1675	The second time constant T_2 of bell-shaped acceleration / deceleration for rapid traverse for each axis

1677	The third time constant T_2 of bell-shaped acceleration / deceleration in rapid traverse for each axis

[Input type] Parameter input
[Data type] Word axis
[Data unit] msec
[Data range] 0 to 512

Specify the second or the third time constant T2 used for bell-shaped of acceleration / deceleration after interpolation for rapid traverse for each axis. When these parameters are set values beyond the range, the specification of 0 is assumed.

1678	The second time constant of acceleration / deceleration for cutting feed for each axis

1679	The third time constant of acceleration / deceleration for cutting feed for each axis

[Input type] Parameter input
[Data type] Word axis
[Data unit] msec
[Data range] -1 to 4000

Specify the second or the third time constant of acceleration / deceleration after interpolation for cutting feed for each axis. As for the axis for which these parameters are set values -1 or beyond the range, the second or the third time constant for cutting feed is invalid.

	#7	#6	#5	#4	#3	#2	#1	#0
25750							BCT	BRT

[Input type] Parameter input
[Data type] Bit path

#0 BRT By time constant of acceleration/deceleration after interpolation for rapid traverse switching signals RTC2, RTC3 <Gn599.6, Gn599.7>, time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse is:
0: not switched.
1: switched.

#1 BCT By time constant of acceleration/deceleration after interpolation for cutting feed switching signals CTC2, CTC3 <Gn599.4, Gn599.5>, time constant of acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode is:
0: not switched.
1: switched.

25751	the second time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse for each axis
25752	the third time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse for each axis

[Input type] Parameter input
[Data type] Word axis
[Data unit] msec
[Data range] -1 to 4000

Set the second or third time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse for each axis.
When the parameter is set to -1, the second or third time constant of acceleration/deceleration after interpolation of acceleration/deceleration before interpolation in rapid traverse is disabled.

25753	the second time constant of acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode for each axis
25754	the third time constant of acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode for each axis

[Input type] Parameter input
[Data type] Word axis
[Data unit] msec
[Data range] -1 to 4000

Set the second or third time constant of acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode for each axis.
When the parameter is set to -1, the second or third time constant of acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode is disabled.

7.2.11 Servo loop gain / in-position width switching function by signal

Overview

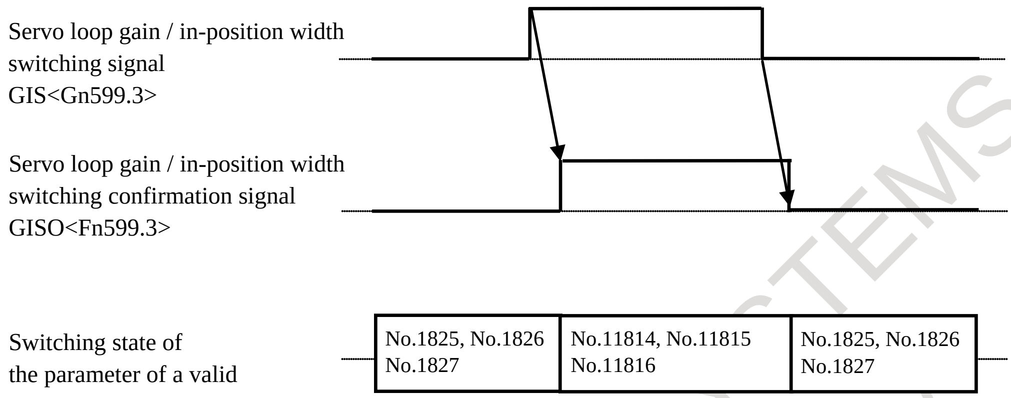
Servo loop gain / in-position width can be switched by input signal of PMC.

Explanation

In this function, servo loop gain and in-position width can be switched to either of the two following kinds by servo loop gain/in-position width switching signal GIS <Gn599.3>.

- The usual servo loop gain (Parameter No. 1825), the usual in-position width (Parameter Nos. 1826 and 1827)
- The second servo loop gain (Parameter No. 11814), the second in-position width (Parameter Nos. 11815 and 11816)

Example of time chart for servo loop gain / in-position width switching



Confirm an effective parameter by servo loop gain / in-position width switching confirmation signal GISO.

Note

1) If -1 is set to parameters No. 11814 or a value not within the valid data range is specified in this parameter, the switching of servo loop gain is invalid in its axis.
2) If -1 is set to parameters No. 11815 or a value not within the valid data range is specified in this parameter, the switching of in-position width is invalid in its axis.
3) If -1 is set to parameters No. 11816 or a value not within the valid data range is specified in this parameter, the switching of in-position width for cutting is invalid in its axis.
4) To switch servo loop gain and in-position width, all the valid axes should stop. When the axis has not stopped, servo loop gain and in-position width are switched after stopping all the valid axes.
5) The following tables show using with other functions.

Table 7.2.11 (a) Using with other functions

Function name	Description
Spindle speed function	Because servo loop gain is not used as for spindle motor, this function becomes invalid.
Positioning by optimum accelerations	When this function is valid, positioning by optimum accelerations is invalid. When this function is switched when positioning by optimum accelerations is valid, this function is valid after stopping all the valid axes.
Cs contour control	In Cs contour control, this function is invalid.
Setting axes for interpolation with a Cs contour control axis	When this function is valid, setting axes for interpolation with a Cs contour control axis is invalid. When this function is switched when servo loop gain is switched by setting axes for interpolation with a Cs contour control axis, servo loop gain of this function is valid after stopping all the valid axes.
Spindle positioning	In spindle positioning, this function is invalid.
Rigid tapping	In the spindle of rigid tapping, this function is invalid. In the drilling axis of rigid tapping, this function is invalid.
Index table indexing	As for the control axis set to parameter No.5510 as index table indexing, this function is invalid.
Electronic gear box (EGB)	In electronic gear box, this function is invalid.

The use example is shown as follows. Only servo loop gain is described in the use example, and when servo loop gain of this function is valid, in-position width of this function is valid, too.

- Positioning by optimum accelerations

In X axis, Y axis, and Z axis, positioning by optimum accelerations is valid axes.

Program	Description
O0001 ;	Optimum accelerations is valid
G00X50.0 ;	
Y50.0 ;	
Z50.0 ;	
Mxx ;	Servo loop gain / in-position width switching signal GIS<Gn599.3>="1"
X100.0 ;	This function is valid, Optimum accelerations is invalid
Y100.0 ;	
Z100.0 ;	
Myy ;	Servo loop gain / in-position width switching signal GIS<Gn599.3>="0"
X150.0 ;	Optimum accelerations is valid
Y150.0 ;	
Z150.0 ;	
M30 ;	

- Cs contour control axis, setting axes for interpolation with a Cs contour control axis

C axis is Cs contour control axis、A axis is setting axes for interpolation with a Cs contour control axis (C axis).

Program	Description
O0002 ;	Usual servo loop gain is valid in A axis
G00A50.0 ;	
A100.0 ;	
Maa ;	Cs contour control mode of C axis turns on
A50.0 C50.0 ;	Setting axes for interpolation with a Cs contour control axis is valid in A axis
A0.0 C100.0 ;	
Mxx ;	Servo loop gain / in-position width switching signal GIS<Gn599.3>="1"
A50.0 ;	In A axis, this function is valid, setting axes for interpolation with a Cs contour control axis is invalid
C50.0 ;	
Myy ;	Servo loop gain / in-position width switching signal GIS<Gn599.3>="0"
A100.0 C0.0 ;	Setting axes for interpolation with a Cs contour control axis is valid in A axis
A50.0 C50.0 ;	
Mbb ;	Cs contour control mode of C axis turns off
A0.0 ;	Usual servo loop gain is valid in A axis
M30 ;	

* As for C axis, this function always is invalid because of the spindle motor.

6) It is not changed to the servo loop gain of this function when there is a control axis by which this function is invalid by the other functions, and there is a possibility not to be able to interpolate correctly.

Signal

Servo loop gain / in-position width switching signal GIS<Gn599.3>

[Classification] Input signal

[Function] It switches servo loop gain and in-position width.

[Operation] - When GIS is "0", it switches to the usual servo loop gain (Parameter No.1825), the usual in-position width (Parameter Nos.1826 and 1827).

- When GIS is "1", it switches to the second servo loop gain (Parameter No.11814), the second in-position width (Parameter Nos.11815 and 11816).

Servo loop gain / in-position width switching confirmation signal GISO<Fn599.3>

[Classification] Output signal

[Function] This signal indicates that the change to servo loop gain / in-position width for this function is valid.

[Output condition] This signal is set to “1” when servo loop gain / in-position width switching function by signal is valid. This signal is set to “0” when servo loop gain / in-position width switching function by signal is invalid.

> **NOTE**
> When the usual servo loop gain / in-position width is not used even if this signal is “1”, servo loop gain / in-position width for this function is invalid.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn599					GIS			

	#7	#6	#5	#4	#3	#2	#1	#0
Fn599					GISO			

Parameter

The parameter used by this function is shown as follows.

11814	The second servo loop gain for each axis

[Input type] Parameter input
[Data type] Word axis
[Data unit] 0.01/sec
[Data range] -1, 1 to 9999

Specify the second servo loop gain for each axis.
As for the axis for which this parameter is set values of -1 or beyond the range, the second servo loop gain is invalid.

11815	The second in-position width for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Data unit] Detection unit
[Data range] -1 to 99999999

Specify the second in-position width for each axis.
As for the axis for which this parameter is set values of -1 or beyond the range, the second in-position width is invalid.

11816	The second in-position width in cutting for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Data unit] Detection unit
[Data range] -1 to 99999999

Specify the second in-position width in cutting for each axis.
As for the axis for which this parameter is set values of -1 or beyond the range, the second in-position width in cutting is invalid.

7.2.12 IN-ACCELERATION/DECELERATION SIGNAL

M

Overview

During deceleration in a block for cutting feed, the in-acceleration/deceleration signal ACDEC<Fn520.3> is output to the machine.

Explanation

If deceleration is performed in a block for cutting feed (G01, G02, G03), the in-acceleration/deceleration signal ACDEC<Fn520.3> is output to the machine during deceleration. For the time set in parameter No. 1650 after the execution of the next block (or the remaining movement of the current block) is started upon completion of deceleration, the ACDEC signal continues to be output. The ACDEC signal is turned off when the set time has elapsed.
In this case, deceleration means any of the cases indicated below.
That is, during execution of a block for cutting feed (G01, G02, G03), deceleration is specified to decrease the speed to 0 by any of the following:

(1) Interlock
(2) Override 0%
(3) Single-block stop
(4) Automatic operation stop by feed hold
(5) Exact stop check
(6) The next block neither specifies cutting feed nor movement.

During deceleration in such a case, the ACDEC is output.

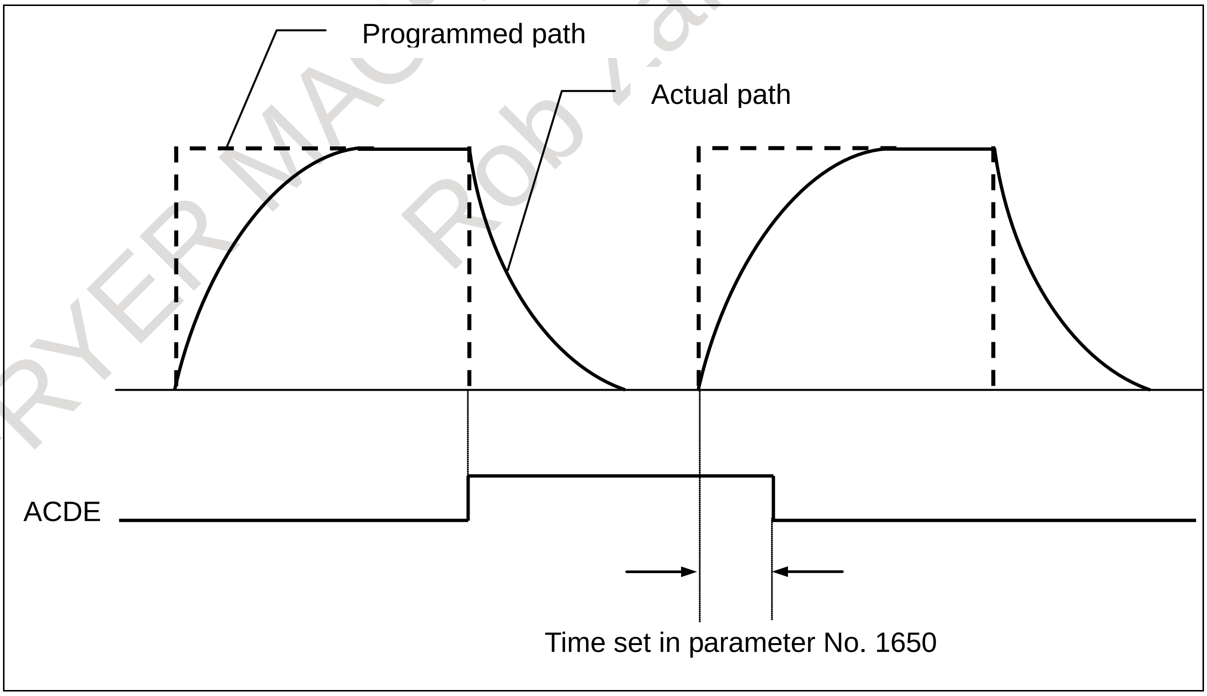


Fig.7.2.12 (a) In-acceleration/deceleration signal

For example, if interlock is applied during cutting feed, the ACDEC signal is output. The ACDEC signal is turned off when the time set in parameter No. 1650 has elapsed after interlock is canceled. If the block after the output of the ACDEC signal is a block not specifying cutting feed, such as a block specifying positioning (G00) or specifying an M function alone, the ACDEC signal continues to be output. In this case, the ACDEC signal is turned off when the time set in parameter No. 1650 has elapsed after execution of the next block for cutting feed is started.

NOTE
The ACDEC signal is turned off by a reset.

Signal

In-acceleration/deceleration signal ACDEC<Fn520.3>

[Classification] Output signal
[Function] Posted to the PMC during deceleration in a block for cutting feed (G01, G02, G03).
[Operation] If deceleration is performed in a block for cutting feed (G01, G02, G03), this signal is set to 1 during deceleration. This signal continues to be set to 1 for the time set in parameter No. 1650 after execution of the next block is started upon completion of deceleration. This signal is set to 0 when the time set in parameter No. 1650 has elapsed.
This signal is also set to 0 by a reset.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn520					ACDEC			

Parameter

1650	Timer for the acceleration/deceleration signal

[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 32767
Set the output duration of the acceleration/deceleration signal.

Note

NOTE
This function is disabled for look-ahead acceleration/deceleration before interpolation.

7.3 JERK CONTROL

M

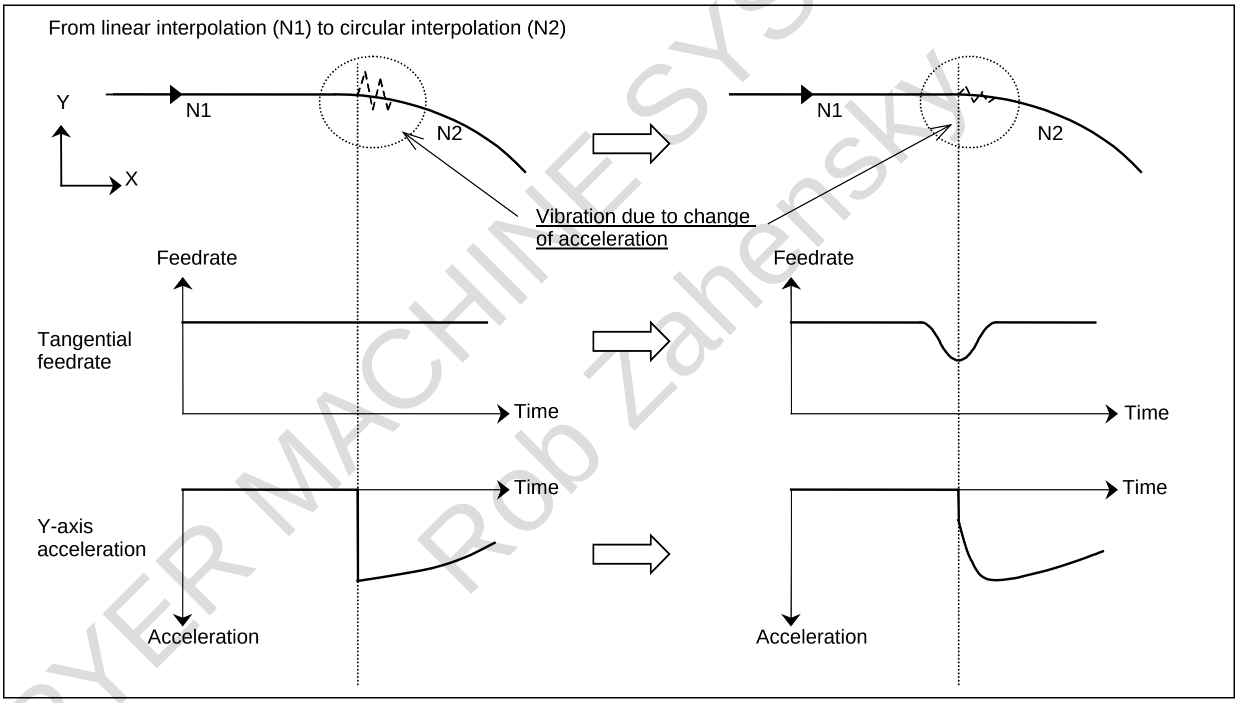
7.3.1 Speed Control with Change of Acceleration on Each Axis

Overview

In portions in which acceleration changes largely, such as a portion where a programmed figure changes from a straight line to curve, vibration or shock on the machine may occur. Speed control with change of acceleration on each axis is a function to suppress machining errors due to vibration and machine shock generated by change of acceleration. This function obtains a feedrate so that change of acceleration is within the parameter-set permissible acceleration change amount for each axis, and performs deceleration by using look-ahead acceleration/deceleration before interpolation.

Explanation

In the following example, the Y-axis acceleration changes largely at the contact point between a linear interpolation and circular interpolation, so deceleration is performed.



- Setting the permissible acceleration change amount

The permissible acceleration change amount for each axis is set in parameter No. 1788. When 0 is set in this parameter for a certain axis, speed control with change of acceleration is not performed for that axis.

- Parameter setting example

Suppose a figure shown Fig. 7.3.5 in which a straight line is followed by an arc. Let the specified feedrate and the arc radius be 6000 mm/min and 10 mm, respectively. Then, the Y-axis acceleration change amount at the contact point of the linear and arc portions is obtained as follows:

$$\frac{v^2}{r} = 1000mm/s^2$$

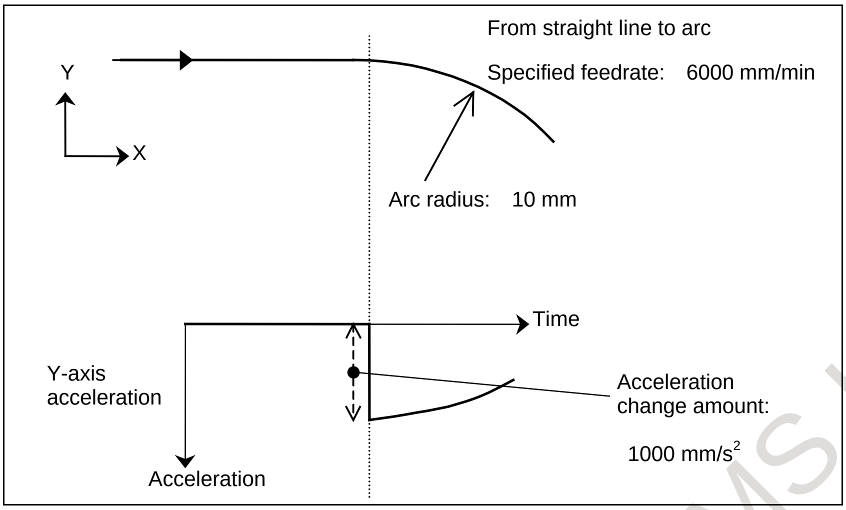
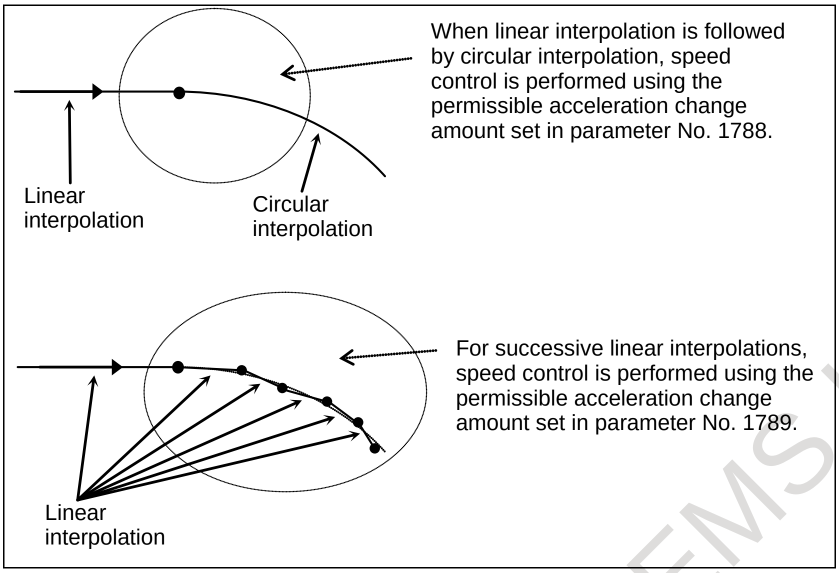


Fig. 7.3.1

To suppress the change of acceleration to 300 mm/sec^2, set 300 mm/sec^2 for the Y-axis in parameter No. 1788.
Note that the change of acceleration is determined from the interpolation data of the CNC, so it may differ from the theoretical value.
The actual machine is affected by acceleration/deceleration and other factors, so the value to be set in the parameter should be determined after adjustments are made.

- For successive linear interpolations

When there are successive linear interpolations, speed control with change of acceleration obtains the deceleration feedrate from the change in acceleration between the start point and end point of a specified block.
When a curve is specified using successive minute straight lines, programmed values are rounded to the least input increment before issued, so the machining profile is approximated with a broken line. The error due to rounding may increase change of acceleration, and especially when the line segments specified by blocks are short, deceleration is performed frequently. As a result, the machining speed cannot increase enough. In such a case, a relatively large value should be set in parameter No. 1789 as the permissible acceleration change amount for each axis in successive linear interpolations to improve the machining speed.
When a value other than 0 is set in parameter No. 1789 for an axis for which deceleration with change of acceleration is enabled, this setting is regarded as the permissible acceleration change amount at corners in which linear interpolations meet. (For portions where a linear interpolation and circular interpolation meet and where circular interpolations meet, the setting in parameter No. 1788 is used.)
When 0 is set in parameter No. 1789 for an axis, the setting in parameter No. 1788 specifying the ordinary permissible acceleration change amount is used even at a corner in which linear interpolations meet.
When smooth speed control is used in speed control with permissible acceleration in AI contour control II, the deceleration feedrate is obtained from the change of acceleration calculated by smooth speed control.
Therefore, the deceleration feedrate may be higher than the ordinary deceleration feedrate.



Parameter

1788	Maximum allowable acceleration change rate in feedrate determination based on acceleration change for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Set a maximum allowable acceleration change rate for each axis in feedrate control based on acceleration change under control on the rate of change of acceleration.
For an axis with 0 set in this parameter, feedrate control based on acceleration change is disabled.
If 0 is set for all axes, feedrate control based on acceleration change is not exercised.

1789	Maximum allowable acceleration change rate in feedrate determination based on acceleration change for each axis (linear interpolation)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Set a maximum allowable acceleration change rate for each axis in feedrate control based on acceleration change under control on the rate of change of acceleration in successive linear interpolation operations.
In feedrate control based on acceleration change at a corner between linear interpolation operations, the maximum allowable acceleration change rate not set in parameter No. 1788 but set in this parameter is valid.
For an axis with 0 set in this parameter, the maximum allowable acceleration change rate set in parameter No. 1788 is valid.
Feedrate control based on acceleration change is disabled for an axis with 0 set in parameter No. 1788, so that the setting of this parameter for such an axis is ignored.

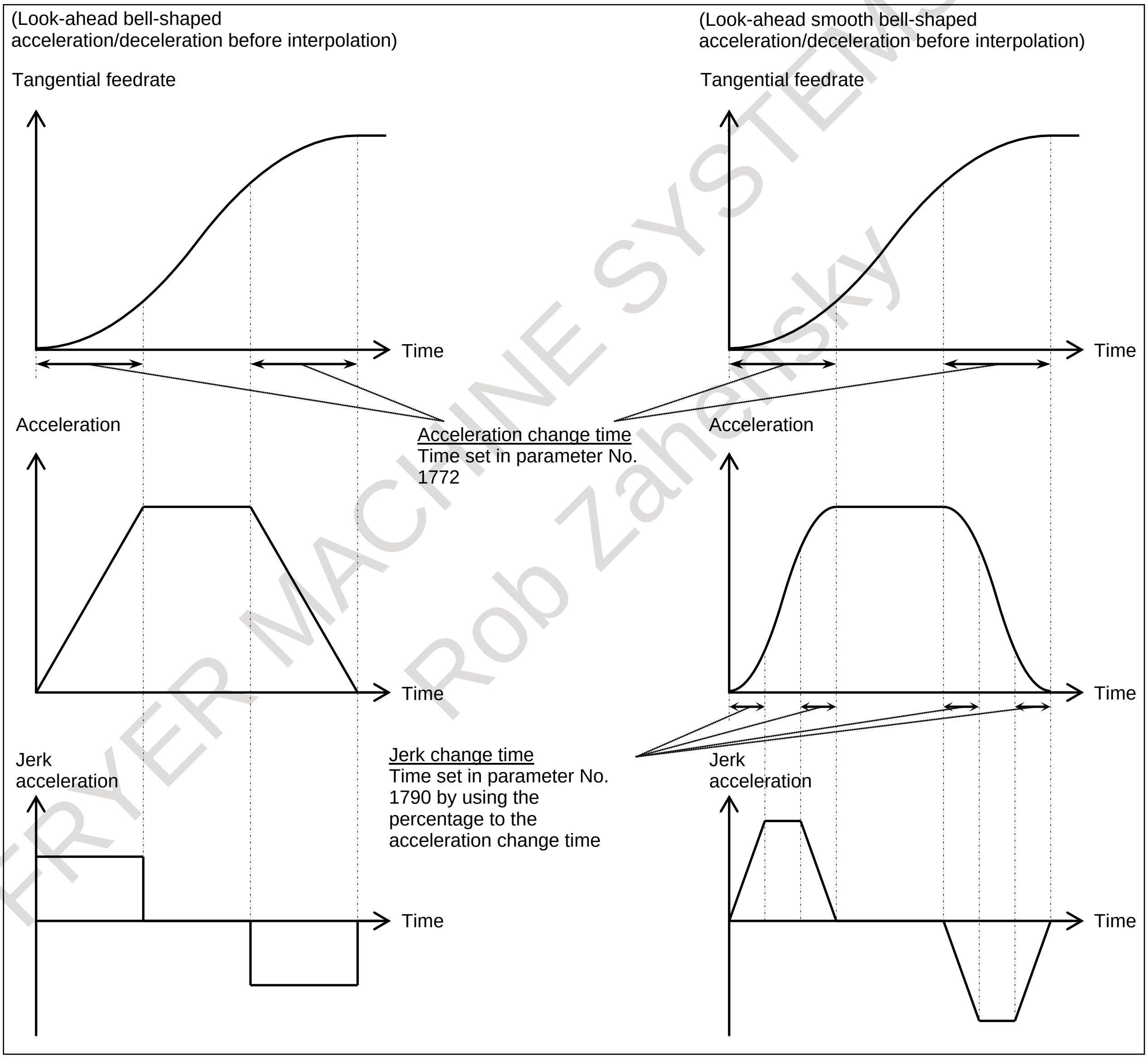
7.3.2 Look-Ahead Smooth Bell-Shaped Acceleration/Deceleration before Interpolation

Overview

In look-ahead bell-shaped acceleration/deceleration before interpolation performs smooth acceleration/deceleration by changing the acceleration at a constant rate in specified acceleration change time.

In look-ahead smooth bell-shaped acceleration/deceleration before interpolation, the jerk change time is specified in parameter No. 1790 by using the percentage to the acceleration change time for look-ahead bell-shaped acceleration/deceleration before interpolation, and change of acceleration is also controlled so that the change is bell-shaped.

This enables smoother acceleration/deceleration, therefore it reduces machine vibration and shock due to acceleration/deceleration.



Explanation

- Setting the jerk change time

The jerk change time is set in parameter No. 1790 by using the percentage to the acceleration change time.
The actual jerk change time is represented by the percentage to the acceleration change time set in parameter No. 1772.
The jerk change time must be within a half of the acceleration change time, so the value to be set in the parameter ranges 0 to 50 (percent).
If 0 or a value beyond the specifiable range is specified in parameter No. 1790, look-ahead smooth bell-shaped acceleration/deceleration before interpolation is not available.

- Acceleration/deceleration before interpolation for linear type rapid traverse

When look-ahead bell-shaped acceleration/deceleration is used in look-ahead acceleration/
deceleration before interpolation for linear type rapid traverse, enabling look-ahead smooth bell-shaped acceleration/deceleration before interpolation applies smooth bell-shaped acceleration/deceleration to acceleration/deceleration before interpolation for linear type rapid traverse.
In this case, the jerk change time is represented by the percentage set in parameter No. 1790 to the acceleration change time set in parameter No. 1672.

- Optimum torque acceleration/deceleration

When bell-shaped acceleration/deceleration is used in optimum torque acceleration/deceleration, enabling look-ahead smooth bell-shaped acceleration/deceleration before interpolation applies smooth bell-shaped acceleration/deceleration to optimum torque acceleration/
deceleration.
In this case, the jerk change time is represented by the percentage set in parameter No. 1790 to the acceleration change time set in parameter No. 1672.

Parameter

1790	Ratio of change time of the rate of change of acceleration in smooth bell-shaped acceleration/deceleration before interpolation

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 50

Set the ratio of the change time of the rate of change of acceleration to the change time of acceleration(*1) by percentage (%) in smooth bell-shaped acceleration/deceleration before look-ahead interpolation.
If 0 is set in this parameter or a value not within the valid data range is specified in this parameter, look-ahead smooth bell-shaped acceleration/ deceleration before interpolation is not performed.

(*1) Parameter No. 1772 for look-ahead acceleration/deceleration before interpolation (cutting feed).
Parameter No. 1672 for look-ahead acceleration/deceleration before interpolation in linear rapid traverse, or for optimum torque acceleration/ deceleration.

8 MULTI-PATH CONTROL

8.1 MULTI-PATH CONTROL

Overview

The multipath control function is designed to control the independent simultaneous machining with up to 2 paths, and the peripheral device such as a loader for performing a non-machining operation. (machining path up to 2, loader path up to 2, up to 4 paths in total). This function is applicable to lathes and automatic lathes which perform cutting simultaneously with multiple tool posts, combined machine tools which perform turning and milling simultaneously with multiple paths, and machines which require additional control paths such as a loader control path.

For multipath simultaneous machining, each machining program is stored in a directory in program memory for each path. When automatic operation is to be performed, each path is activated after selecting a program for machining with path 1 and programs for machining with paths 2 from the programs stored in the respective directories in program memory. Then the programs selected for the tool posts are executed independently at the same time. When tool post 1 and tool post 2 need to wait for each other during machining, the waiting function is available.

Other available functions specific to multipath control include interference check for each path, balance cut, synchronization control, composite control, path spindle control, and memory common to paths.

Just one LCD/MDI is provided for the all paths. Before operation and display on the LCD/MDI, the path selection signal is used to switch between the paths.

- For a system with two paths

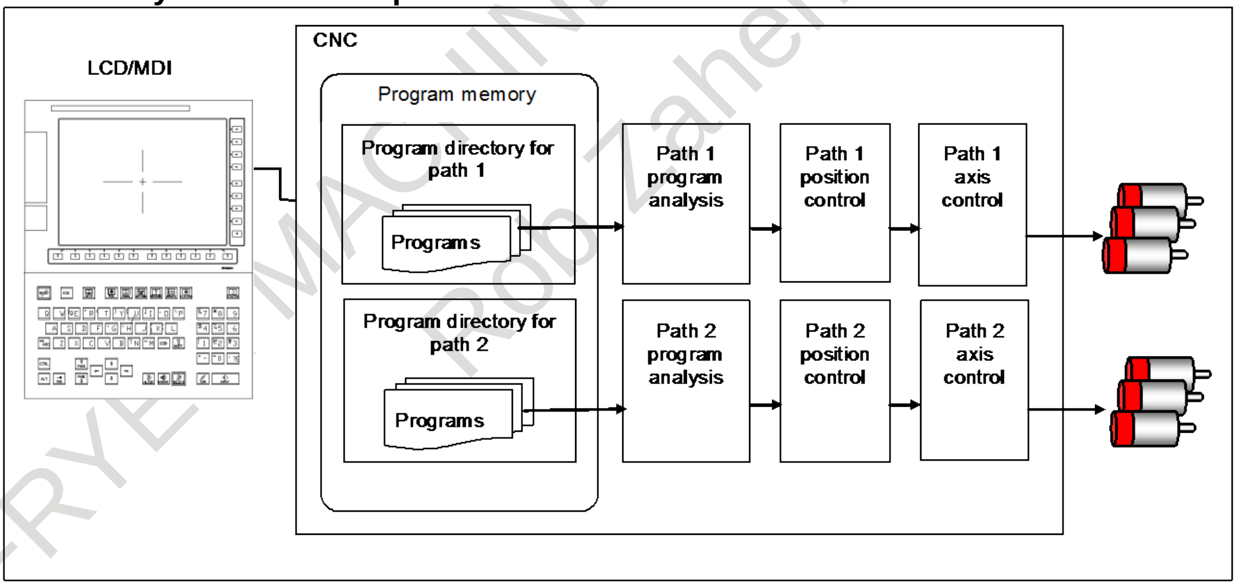


Fig.8.1 (a)

Explanation

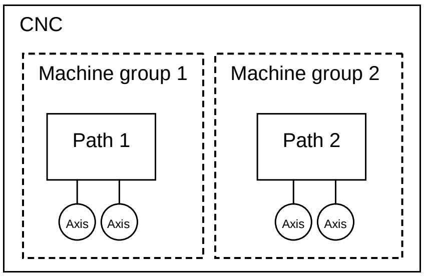
- System configuration

In multi-path control, multiple paths are combined into a single machine group, and such multiple machine groups, in turn, make up a single system.

Each of the elements making up multi-path control is explained below.

Example)

Configuration of multi-path control



- Machine group

If you have multiple paths, you can combine some of them into a group, so that, for example, you can share data in that group and if an alarm is generated in a certain path, you can stop other paths in that group. A group that is made up of such paths is called a machine group.
Up to three groups are available, depending on the type of the CNC system.
The main items that depend on the machine group are shown below.

- Emergent stop signal *ESP <X0008.4,X0008.0,X0008.1>
- MDI reset key
- Operation during the occurrence of an alarm

- Path

A path refers to a group of axes controlled by the same CNC program command.
Up to 2 paths are available for machining, and up to 2 paths are available for loader control. Up to 4 paths in total are available.
You can specify which machine group a certain path is to belong to, using parameter No. 0980 for that path. In addition.

NOTE
The number of the machine group to which each path can belong differs depending on the machine used. For details, refer to the manual issued by the machine tool builder.

- Controlled axes

The number of control axes witch used throughout the entire system is specified with parameter No.0987.
You can specify which path each controlled axis is to be assigned, using parameter No. 0981.
You can assign each path to any controlled axis, provided that you follow the maximum allowable number of controlled axes.

Example)
For a system with 2 paths and a total of 6 controlled axes:
Settings that cause the system to become a 2-path system with the first path consisting of 3 controlled axes and the second consisting of 3 controlled axes

Parameter No. 0981	Setting value	Application
Logical, axis 1	1	Path 1, axis 1
Logical, axis 2	1	Path 1, axis 2
Logical, axis 3	1	Path 1, axis 3
Logical, axis 4	2	Path 2, axis 1
Logical, axis 5	2	Path 2, axis 2
Logical, axis 6	2	Path 2, axis 3

> **NOTE**
> 1 If the setting of parameter No. 0981 is 0, that controlled axis will belong to the first path.
> 2 Any path having no controlled axis cannot be set.

- Spindle

The number of spindle which used throughout the entire system is specified with parameter No.0988.
You can specify which path each spindle is to be assigned, using parameter No. 0982.
You can assign each path to any controlled axis, provided that you follow the maximum allowable number of spindles.
Example)

For a system with 2 paths and a total of 6 spindles:
Settings that cause the system to become a 2-path system with the first path consisting of 3 spindles and the second consisting of 3 spindles

Parameter No. 0982	Setting value	Application
Logical, spindle 1	1	Path 1, spindle 1
Logical, spindle 2	1	Path 1, spindle 2
Logical, spindle 3	1	Path 1, spindle 3
Logical, spindle 4	2	Path 2, spindle 1
Logical, spindle 5	2	Path 2, spindle 2
Logical, spindle 6	2	Path 2, spindle 3

> **NOTE**
> If the setting of parameter No. 0982 is 0, that spindle will belong to the first path.

Example

- Example of a system configuration (2 path and loader path)

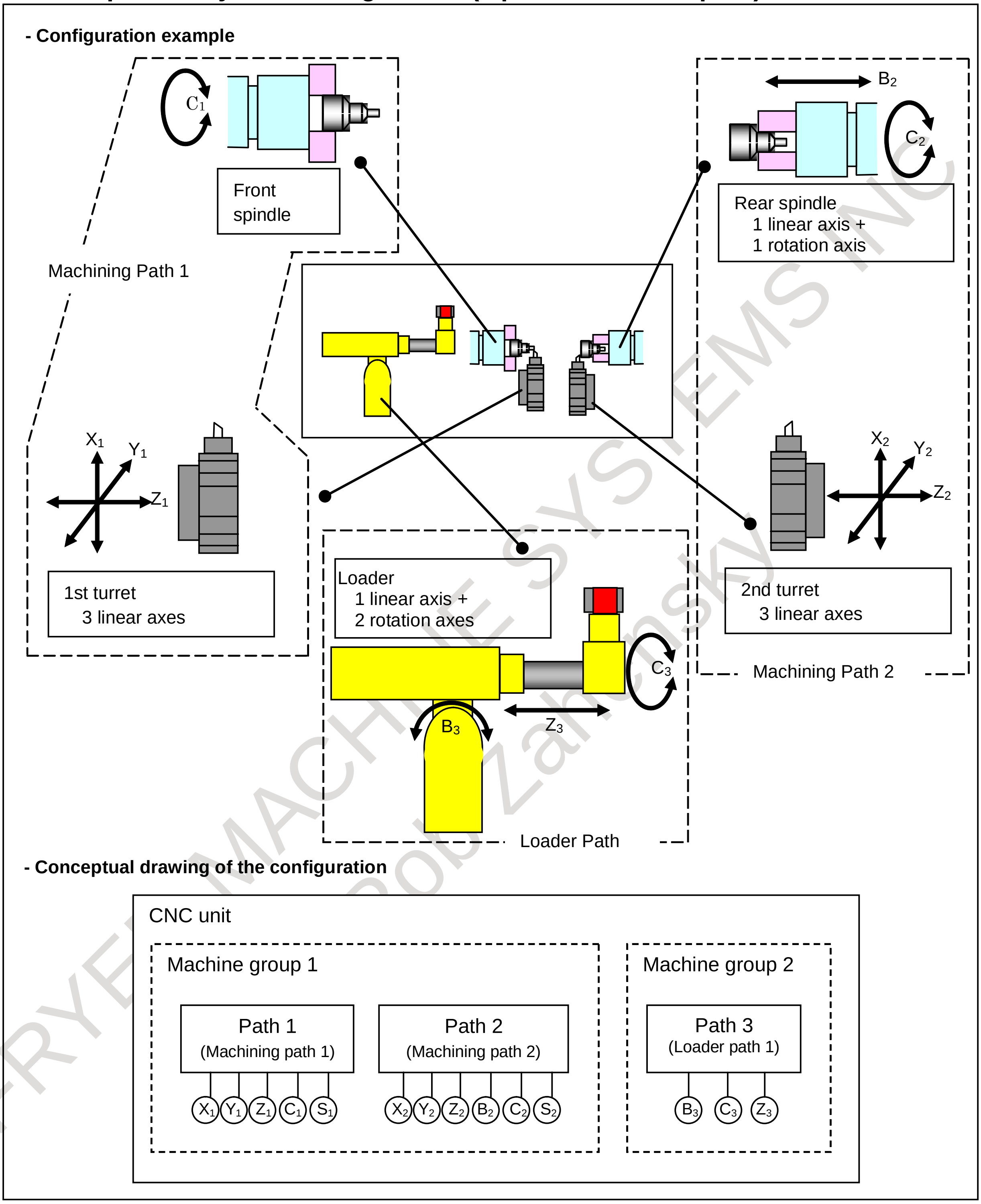


Fig. 8.1 (b)

In the system configuration example shown in the example, Fig. 8.1 (a), paths 1 and 2 are of lathe systems, path 3 is of a machining center system, and path 4 is the loader control path in a lathe system. The machine control type is com y ined system. The parameter settings that accomplish that system configuration are as follows.

Machine Group to Which Each Path Is to Belong (Parameter No. 0980)

	Setting value
Path 1	1
Path 2	1
Path 3 (Loader control)	2

Number of the Path to Which Each Axis Is to Belong (Parameter No. 0981)

	Setting value
Logical, axis 1 (X_1)	1
Logical, axis 2 (Y_1)	1
Logical, axis 3 (Z_1)	1
Logical, axis 4 (C_1)	1
Logical, axis 5 (X_2)	2
Logical, axis 6 (Y_2)	2
Logical, axis 7 (Z_2)	2
Logical, axis 8 (B_2)	2
Logical, axis 9 (C_2)	2
Logical, axis 10 (X_3)	3
Logical, axis 11 (Y_3)	3
Logical, axis 12 (Z_3)	3

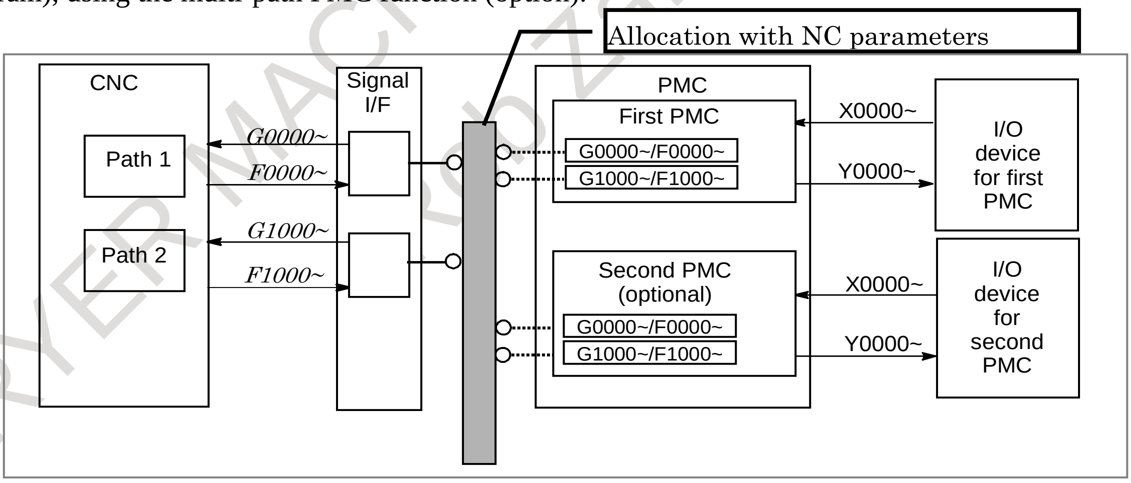
Number of the Path to Which Each Spindle Is to Belong (Parameter No. 0982)

	Setting value
Logical, spindle 1 (S_1)	1
Logical, spindle 2 (S_2)	2

- DI/DO

In a multi-path system, the interface signal address relations between the CNC and the PMCs are as shown in the figure.

You can control all paths with a single PMC (Ladder program), or control each path with a PMC (Ladder program), using the multi-path PMC function (option).



NOTE
1. In a multi-path PCM system, each PMC has an independent signal area. The F, G, X, and Y signal addresses of each PMC are allocated, starting with address 0. In contrast, the F and G signal addresses as seen from the CNC are fixed addresses in the same order as that of the path numbers. The F and G signal addresses programmed by each Ladder program differ from the F and G signal addresses as seen from the CNC.
2. The signal addresses relations between the CNC and the PMCs of each path can be set with CNC parameters Nos. 11920 to 11929. For details of how to do this, refer to Chapter 1 of the PMC Programming Manual.

Specific Setting Example for the Previous Example

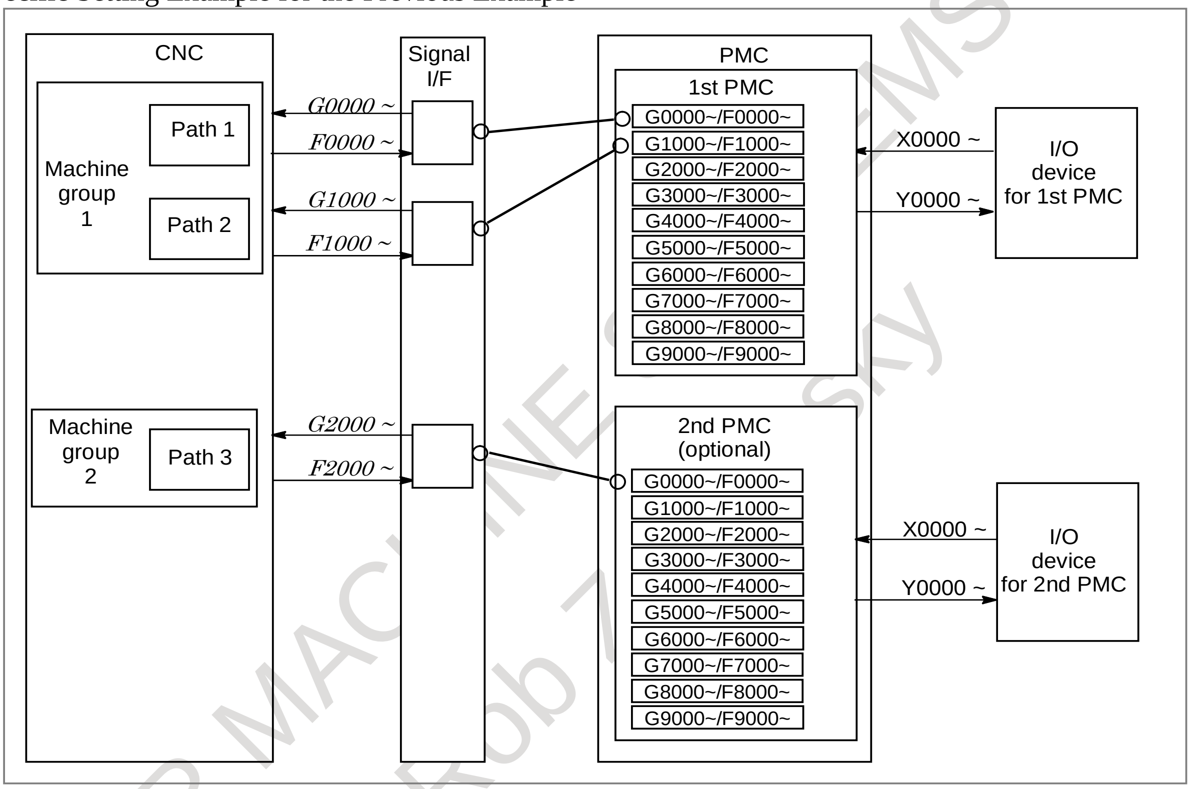


Fig. 8.1 (c)

NOTE
As with the loader control function of the Series 16*i*/18*i*/21*i*, if the PMC (Ladder program) specifically for loaders is used, assign a single PMC path as the loader path, using the multi-path PMC function as in the Fig. 8.1 (c).

Setting Examples for the Interface Between the NC and the PMCs

Parameter No.	Setting value	Application
11920	100	F0000 to F0767/G0000 to G0767 of the 1st PMC
11921	101	F1000 to F1767/G1000 to G1767 of the 1st PMC
11922	102	F2000 to F2767/G2000 to G2767 of the 1st PMC
11923	200	F0000 to F0767/G0000 to G0767 of the 2nd PMC
11924	0	
11925	0	
11926	0	
11927	0	
11928	0	

Parameter No.	Setting value	Application
11929	0	

In general, the address for each path as seen from the NC section is assigned as in the table below.

Signal Address for Each Path

Signal address	Description
G0000 to G0767	Signal for path 1 (PMC→CNC)
F0000 to F0767	Signal for path 1 (CNC→PMC)
G1000 to G1767	Signal for path 2 (PMC→CNC)
F1000 to F1767	Signal for path 2 (CNC→PMC)

The following signals follow separate settings from the above:
- Signals common to paths. (There are those arranged on path 1.)
- Signals for each controlled axis and for each spindle
- Signals for each group in PMC axis control

- Signals common to paths

These signals are common to paths.

Path select

Path select signal HEAD<G0063.0>

Auto screen clear function

Auto screen clear invalidation signal *CRTOF<G0062.1>

Screen hard copy function

Hard copy cancellation request signal HCABT<G0067.6>
Hard copy cancellation request acceptance signal HCAB2<F0061.2>
Hard copy execution request signal HCREQ<G0067.7>
Hard copy under execution signal HCEXE<F0061.3>

External key input function

External key input mode select signal ENBKY<G0066.1>
Key code read signal EKSET<G0066.7>
Key code signal EKC0 to EKC7<G0098.0to G0098.7>
Program screen display in progress signal PRGDPL<F0053.1>
Key code read completion signal EKENB<F0053.7>
Key input nullification signal INHKY<F0053.0>

NOTE
The signal area is common to paths. Key input is made on the currently selected screen.

Memory protection function

Memory protection signal KEY1 to KEY4<G0046.3 to G0046.6>
Memory protect signal KEYO<F0075.6>

8.1.1 CNC Data Display, Setup, and Input/Output

There are offsets and custom macro variables for each path. For the path selected from the display unit, you can perform data display, setup, and input/output using a medium such as a memory card.
As for custom macro variables, you can change some or all of common variables into those for common use by paths, through appropriate parameter settings.
As for offsets, you can change them into those for common use by paths, through appropriate parameter settings.
As for input/output, you can rename the files to be input and output for each path, through appropriate parameter settings.

8.1.2 Multi-path Functions

The functions specific to multi-path control are as listed below.

Waiting M codes
Path interference check
Balance cutting
Synchronous/Composite control
Superimposed control
Path spindle control
Memory common to paths
Path single block check function
Path selection/arbitrary path name display

NOTE
The functions waiting M code, memory common to paths, and path selection/arbitrary path name display are valid for all machine groups. For example, waiting can be performed across machine groups 1 and 2. The other functions are valid only within the same machine group. For example, synchronous control cannot be performed across machine groups 1 and 2.

The following provides an outline of each of the functions.

- Waiting M codes

You use an M code if you want to achieve synchronization in the middle of the machining program for a path.
By specifying a waiting M code in the machining program for each of multiple paths, you can synchronize these paths. If, during automatic operation, a waiting M code is specified for one path, the program waits for the same M code to be specified for another path, then starts executing the next block.
You must specify the range of M codes that can be used as waiting M codes in advance, using an appropriate parameter.
For details, see Section, "WAITING M CODES".

T

- Path interference check

If you machine a single workpiece on two tool posts at the same time, the portions of these tool posts may come close to each other exceedingly. If these two tool posts come into contact due to a program error or other setting error, this will cause grave consequences such as the damage of the tool or the damage of the machine itself.
The "Interference check for each path" refers to the function whereby if a command that will cause two tool posts to interfere with each other as described above is specified, these two tool posts are both decelerated to a stop before they actually come into contact.
For details, see Section, "INTERFERENCE CHECK FOR EACH PATH".

T

- **Balance cutting**

If turning a thin workpiece, you can machine it from both of its sides at the same time by using a cutting tool, thereby preventing the workpiece from deflection, which can occur if it is machined from a single side only, and achieving higher precision. Unless the movements of the two tools are synchronized with each other well, the workpiece will vibrate, making correct machining impossible. By using this function, you can easily synchronize the movements of the tool posts and execute machining.
For details, see Section, "BALANCE CUTTING" in this chapter.

- **Synchronous/Composite control**

Usually, in 2-path control, the tool moves along the axes (X1, Z1, and C1) belonging to tool post 1 in accordance with the move commands for tool post 1 and along the axes (X2, Z2, and C2) belonging to tool post 2 in accordance with the move commands for tool post 2 (independent control for each tool post). The synchronous/composite control function allows you to move the tool along any of the axes of one tool post in synchronization with any of the axes of the other tool post (synchronous control). The function also allows you to replace the move command for any of the axes of one tool post and that for anys of the axes of the other tool post with other to move the tool along these axes with these commands (composite control).
For details, see Section, "SYNCHRONOUS/COMPOSITE CONTROL" in this chapter.

- **Superimposed control**

Usually, in 2-path control, the tool moves along the axes (X1, Z1, and C1) belonging to tool post 1 in accordance with the move commands for tool post 1 and along the axes (X2, Z2, and C2) belonging to tool post 2 in accordance with the move commands for tool post 2 (independent control for each tool post). The superimposed control function allows you to move the tool along any of the axes of one tool post and along any of the axes of the other tool post by superimposing the amount of travel.
For details, see Section, "SUPERIMPOSED CONTROL" in this chapter.

- **Path spindle control**

This function allows the spindle belonging to one path to follow the spindle command of another specific path.
It allows you to specify that a path is to incorporate the position coder feedback data for any of the spindles belonging to any path. This achieves control using a spindle of another path, for example, threading or feed per revolution in path 1, using a spindle belonging to path 2.
For details, see Section, "PATH SPINDLE CONTROL".

- **Memory common to paths**

In a machine with multiple paths, this function allows you to make common the independent custom macro common variables and tool compensation memory of a path, through appropriate parameter settings.

- **Custom macro common variables**

Part or all of common variables #100 to #149 and #500 and #531 of custom macros can be made into those for common use by paths (variables that permit reading or writing by either of both paths).

- **Tool compensation memory**

The tool compensation memory of path 1 can be accessed by path 2 as well, for data reference and setting.
The requirement is that the settings for tool compensation (such as the number of sets, the number of digits, and the unit system) must be the same in both paths 1 and 2.

For details, see Section, "MEMORY COMMON TO PATHS" in this chapter.

- Path single block check function

If one path enters the single block stop state, this function allows another path to enter the feedhold stop state, enabling single block operation with the path machining programs being nearly synchronized.
For details, see the Section, "PATH SINGLE BLOCK CHECK FUNCTION".

- Path selection/arbitrary path name display

Path selection/arbitrary path name display
This function is used to select a path targeted for an operation that is performed for a specified path, for example, displaying and setting various types of data (tool compensation or the like) for each path, inputting the command program in MDI mode, or editing the machining program in program memory.
In addition, the name of each path can be changed by setting the parameter.
For details, see Section, "PATH SELECTION/ARBITRARY PATH NAME DISPLAY" in this chapter.

8.1.3 Cautions on Multi-path Control

- Parameter

> ⚠ **WARNING**
> If you attempt to change parameters in MDI mode in a certain path, it is probable that other paths are in MEM or other modes even if that path is in MDI mode. Before changing parameters, check to see if this will not affect the operations in the other paths.
> During axial movement, in particular, use great caution not to change those parameters related to axes and spindles.

- MDI reset key

> ⚠ **CAUTION**
> Only a single LCD/MDI set is provided for all paths. You use the path selection signal to specify which path the operations and displays on the LCD/MDI are for. The RESET key on the MDI panel is, however, effective to all paths, regardless of the path selection signal.
> Note, however, that you can make the key effective to the path selected with the path selection signal or the machine group, by appropriately setting bit 0 (RST) of parameter No. 8100 or bit 0 (MGR) of parameter No. 8106).

- Alarms during automatic operation

> ⚠ **CAUTION**
> If, during automatic operation, an alarm occurs in a certain path, all paths in that machine group will enter the feedhold state and stop.
> You can let them continue to operate, disregarding the alarm, by appropriately setting bit 1 (IAL) of parameter No. 8100.

- Assignment of signals

> ⚠ **CAUTION**
> 1 When there are four or more paths
> Input signals by the X address for each path are assigned to up to three paths. Therefore, when there are four or more paths, the signals must be assigned by bit 2 (XSG) of parameter No. 3008.

⚠ CAUTION

2 When there are nine or more axes for one path
The X address of the reference position return deceleration signal (*DECx) for each axis is assigned to up to three paths (up to eight axes for each path). Therefore, when there are four or more paths or there are nine or more axes for one path, the signals must be assigned by setting bit 2 (XSG) of parameter No. 3008, parameter No. 3013, and parameter No. 3014.
The input signal (G signal) and output signal (F signal) for each axis are assigned to up to eight axes for each path. Therefore, when there are nine or more axes for one path, the signals must be assigned by setting parameter No. 3021.

Parameter

0980	Machine group number to which each path belongs

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 3
Set the machine group number to which each path belongs.

NOTE
When 0 is set, each path is assumed to belong to machine group 1.

0981	Absolute path number to which each axis belongs

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 1 to 4
Set the path to which each axis belongs.

NOTE
When 0 is set, each axis is assumed to belong to path 1.

0982	Absolute path number to which each spindle belongs

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 1 to 4
Set the path to which each spindle belongs.

NOTE
When 0 is set, each spindle is assumed to belong to path 1.

0983	Path control type of each path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 1
Set the path control type of each path.
The following two path control types are available:
T series (lathe system) : 0
M series (machining system): 1

	#7	#6	#5	#4	#3	#2	#1	#0
3008						XSG		

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#2 XSG A signal assigned to an X address is:
0: Fixed at the address.
1: Able to be reassigned to an arbitrary X address.

NOTE
When this parameter is set to 1, set parameters Nos. 3013, 3014, 3012, and 3019. If parameters Nos. 3013 and 3014 are not set, the deceleration signal for reference position return is assigned to bit 0 of <X0000>. If parameters Nos. 3012 and 3019 are not set, the skip signal, the PMC axis control skip signal, the measurement position arrival signal, the interlock signal for each axis direction, and the tool compensation value write signal are assigned to <X0000>.

3012	Skip signal assignment address

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 727
Set an X address to which the skip signal SKIPn is to be assigned.

NOTE

This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
Depending on the configuration of the I/O Link, the actually usable X addresses are:
<X0000 to X0127>, <X0200 to X0327>, <X0400 to X0527>, <X0600 to X0727>

3013	X address to which the deceleration signal for reference position return is assigned

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 727

Set an address to which the deceleration signal *DECx for reference position return for each axis is to be assigned.

NOTE

This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
Depending on the configuration of the I/O Link, the actually usable X addresses are:
<X0000 to X0127>, <X0200 to X0327>, <X0400 to X0527>, <X0600 to X0727>

3014	Bit position of an X address to which the deceleration signal for reference position return is assigned

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

Set a bit position to which the deceleration signal for reference position return *DECx for each axis is to be assigned.

NOTE

This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.

3019	Address to which the PMC axis control skip signal and measurement position arrival signals are assigned

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 727

Set an X address to which the PMC axis control skip signal ESKIP and the measurement position arrival signals (XAE, YAE, and ZAE (Machining Center series) or XAE and ZAE (Lathe series)) are to be assigned.

Example 1. When No.3012 is set to 5 and No.3019 is set to 6

When bit 2 (XSG) of parameter No. 3008 is 1, the PMC axis control skip signal, and measurement position arrival signal are allocated to X0006 and the skip signal is allocated to X0005.

	#7	#6	#5	#4	#3	#2	#1	#0	
X005	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	(T series)
	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	(M series)

	#7	#6	#5	#4	#3	#2	#1	#0	
X006		ESKIP					ZAE	XAE	(T series)
		ESKIP				ZAE	YAE	XAE	(M series)

Example 2. When No.3012 is set to 5 and No.3019 is set to 5

When bit 2 (XSG) of parameter No. 3008 is 1, the PMC axis control skip signal, measurement position arrival signal, and skip signal are allocated to X0005.

	#7	#6	#5	#4	#3	#2	#1	#0	
X005	SKIP	ESKIP					ZAE	XAE	(T series)
		SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	

	#7	#6	#5	#4	#3	#2	#1	#0	
	SKIP	ESKIP	SKIP5	SKIP4	SKIP3	ZAE	YAE	XAE	(M series)
		SKIP6				SKIP2	SKIP8	SKIP7	

NOTE
This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
Depending on the configuration of the I/O Link, the actually usable X addresses are:
X0000 to X0127, X0200 to X0327, X0400 to X0527, X0600 to X0727

3021	Address to which an axis signal is assigned

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7, 10 to 17, 20 to 27, ... , 90 to 97

For each axis of the CNC, set a PMC interface address.
Set a value according to the tables below.

Value of parameter No. 3021 (the second digit)

Setting value	Input signal address	Output signal address
0	G0000 to G0767	F0000 to F0767
1	G1000 to G1767	F1000 to F1767
2	G2000 to G2767	F2000 to F2767
3	G3000 to G3767	F3000 to F3767

Value of parameter No. 3021 (the first digit)

Setting value	Input signal address	Output signal address
0	#0	#0
1	#1	#1
	:	
7	#7	#7

[Example of setting]

Axis number	No. 3021	Signal allocation
1	0	+J1<G0100.0>, -J1<G0102.0>, ZP1<F0090.0>, ...
2	1	+J2<G0100.1>, -J2<G0102.1>, ZP2<F0090.1>, ...
3	2	+J3<G0100.2>, -J3<G0102.2>, ZP3<F0090.2>, ...
4	10	+J4<G1100.0>, -J4<G1102.0>, ZP4<F1090.0>, ...
5	11	+J5<G1100.1>, -J5<G1102.1>, ZP5<F1090.1>, ...

If eight or less axes are used per path, the following signal allocation results when 0 is set for all axes:

Axis 1 of path 1 = Setting equivalent to 0
Axis 2 of path 1 = Setting equivalent to 1
:
Axis 1 of path 2 = Setting equivalent to 10
:

> **NOTE**
> Set this parameter when more than eight axes are used per path.
> The valid data range varies, depending on the system software.

	#7	#6	#5	#4	#3	#2	#1	#0
8100		DSB					IAL	RST

[Input type] Parameter input
[Data type] Bit machine group

#0 RST The pressing of the RESET key on the MDI panel is:

0: Valid for all paths.
1: Valid only for the path selected by the path selection signal.

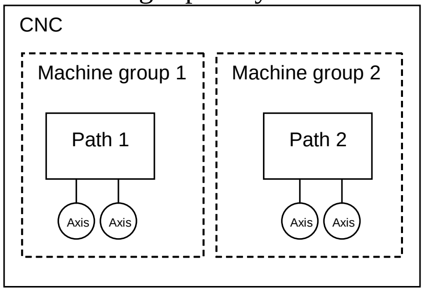
The RESET key on the MDI panel functions for all machine groups. So, in machine groups for which this parameter is set to 0, a reset can be performed for all paths. In machine groups for which this parameter is set to 1, a reset can be performed only for the path that is selected by the path select signal.

> **NOTE**
> The path by which reset becomes actually effective depends on the combination with bit 0 (MGR) of parameter No.8106 setting and this parameter.

So, operations are performed as indicated in the example below.

[Example]

Suppose that the following 3-path system is used.



(1) When RST (machine group 1) = 0 and RST (machine group 2) = 0 are set, the reset key is valid for all paths.

Reset operation when the MDI key is pressed

Signal HEAD	0 (Path 1 selected)	1 (Path 2 selected)
Path 1	Resetting valid	Resetting valid
Path 2	Resetting valid	Resetting valid

(2) When RST (machine group 1) = 1 and RST (machine group 2) = 1 are set, the reset key is valid for a path selected by the path selection signal.

Reset operation when the MDI key is pressed

Signal and HEAD	0	1
Path 1	Resetting valid	Resetting invalid
Path 2	Resetting invalid	Resetting valid

(3) When RST (machine group 1) = 1 and RST (machine group 2) = 0 are set, the following operation is performed:

Reset operation when the MDI key is pressed

Signal HEAD	0	1
Path 1	Resetting valid	Resetting invalid
Path 2	Resetting valid	Resetting valid

#1 IAL Choice of an option concerning operation continuation when an alarm is issued, and choice of an option concerning the start of automatic operation in alarm state:

0: - When an alarm is issued, the operation is stopped with the other path(s) in same group placed in hold state.
- When the other path or paths in same group are placed in alarm state, automatic operation cannot be started.

1: - Even when an alarm is issued, the operation is continued without stopping the other path(s).
- Even when the other path or paths in same group are placed in alarm state, automatic operation can be started.

#6 DSB The inter-path single block check function is:

0: Disabled.
When a single block stop occurs with a path, no single block stop occurs with the other path(s).

1: Enabled.
When a single block stop occurs with a path, a feed hold stop occurs with all paths in the same machine group.

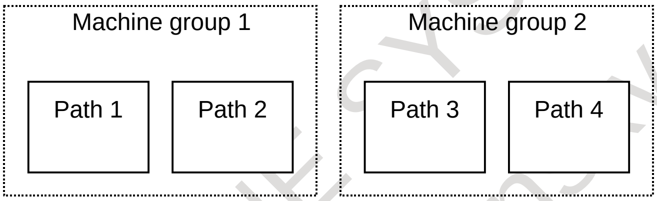
	#7	#6	#5	#4	#3	#2	#1	#0
8106								MGR

[Input type] Parameter input
[Data type] Bit

 # 0 MGR When the RESET key on the MDI panel is pressed,
0: All machine groups are reset.
1: Only the machine groups to which the path selected by the path selection signal belongs are reset.

> **NOTE**
> The path by which reset becomes actually effective depends on the combination with bit 0 (RST) of parameter No. 8100 setting and this parameter.

[Example]
Suppose that the following 2-machine-group with 4-path system is used.



[Example 1]
When the bit 0 (MGR) of parameter No. 8106 is set to 0:
When the RESET key on the MDI panel is pressed, all paths are reset regardless of which path is selected.

[Example 2]
When the bit 0 (MGR) of parameter No. 8106 is set to 0 and path 1 is selected:
When the RESET key on the MDI panel is pressed, machine group 1 (paths 1 and 2) to which path 1 belongs is reset. Machine group 2 (paths 3 and 4) is not reset.

Bit 0 (MGR) of parameter No. 8106 can be used together with bit 0 (RST) of parameter No. 8100. The following table lists the path or paths to be reset when the RESET key on the MDI panel is pressed, according to the setting of bit 0 (RST) of parameter No. 8100 used when bit 0 (MGR) of parameter No. 8106 is set to 1 for the machine group in the above example.

No.	Bit 0 (RST) of parameter No. 8100 (Machine group 1)	Bit 0 (RST) of parameter No. 8100 (Machine group 2)	Selected path			
			Path 1	Path 2	Path 3	Path 4
			Path(s) to be reset			
1	0	0	1, 2	1, 2	3, 4	3, 4
2	0	1	1, 2	1, 2	3	4
3	1	0	1	2	3, 4	3, 4
4	1	1	1	2	3	4

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X0004 T series	SKIP#1	ESKIP#1 SKIP6#1	-MIT2#1 SKIP5#1	+MIT2#1 SKIP4#1	-MIT1#1 SKIP3#1	+MIT1#1 SKIP2#1	SKIP8#1	SKIP7#1
X0004 M series	SKIP#1	ESKIP#1 SKIP6#1	SKIP5#1	SKIP4#1	SKIP3#1	SKIP2#1	SKIP8#1	SKIP7#1
X0007	*DEC8#2	*DEC7#2	*DEC6#2	*DEC5#2	*DEC4#2	*DEC3#2	*DEC2#2	*DEC1#2
X0008				*ESP			(*ESP)	(*ESP)
X0009	*DEC8#1	*DEC7#1	*DEC6#1	*DEC5#1	*DEC4#1	*DEC3#1	*DEC2#1	*DEC1#1
X0010	*DEC8#3	*DEC7#3	*DEC6#3	*DEC5#3	*DEC4#3	*DEC3#3	*DEC2#3	*DEC1#3
X0011 T series	SKIP#3	ESKIP#3 SKIP6#3	-MIT2#3 SKIP5#3	+MIT2#3 SKIP4#3	-MIT1#3 SKIP3#3	+MIT1#3 SKIP2#3	SKIP8#3	SKIP7#3
X0011 M series	SKIP#3	ESKIP#3 SKIP6#3	SKIP5#3	SKIP4#3	SKIP3#3	SKIP2#3	SKIP8#3	SKIP7#3
X0013 T series	SKIP#2	ESKIP#2 SKIP6#2	-MIT2#2 SKIP5#2	+MIT2#2 SKIP4#2	-MIT1#2 SKIP3#2	+MIT1#2 SKIP2#2	SKIP8#2	SKIP7#2
X0013 M series	SKIP#2	ESKIP#2 SKIP6#2	SKIP5#2	SKIP4#2	SKIP3#2	SKIP2#2	SKIP8#2	SKIP7#2

Alarm and message

Number	Message	Description
PS0365	TOO MANY MAXIMUM SV/SP AXIS NUMBER PER PATH	The maximum control axis number or maximum control spindle number which could be used within a path was exceeded.
PW0001	X-ADDRESS(*DEC) IS NOT ASSIGNED.	The X address of the PMC could not be assigned correctly. This alarm may occur in the following case: - During the setting of parameter No. 3013, the X address could not be assigned correctly for the deceleration dog (*DEC) for a return to the reference position. When there are four or more paths or there are nine or more axes for one path, the signals must be assigned by setting bit 2 (XSG) of parameter No. 3008, parameter No. 3013, and parameter No. 3014.
PW0002	PMC address is not correct(AXIS).	The address to assign the axis signal is incorrect. This alarm may occur in the following case: - The parameter No. 3021 setting is incorrect.

8.2 WAITING M CODES

Overview

Control based on M codes is used to cause one path to wait for the other during machining. When an M code for waiting is specified in a block for one path during automatic operation, the other path waits for the same M code to be specified before staring the execution of the next block.

A range of M codes used as M codes for waiting is to be set in the parameters Nos. 8110 and 8111 beforehand. Waiting can be ignored using a signal.

Address P to be specified in the same block as that containing a waiting M code can be set in either of two ways:

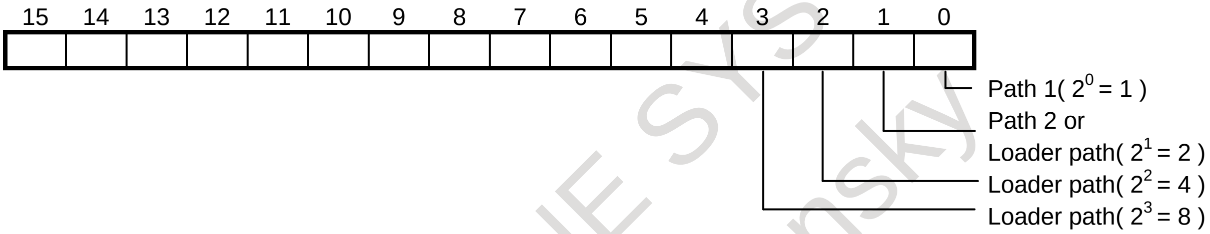
- When the bit 1 (MWP) of parameter No. 8103 is set to 0 :
  Waiting with a binary value specification
- When the bit 1 (MWP) of parameter No. 8103 is set to 1 :
  Waiting with a path number combination specification

- Waiting specified with binary values

When bit 1 (MWP) of parameter No. 8103 is set to 0, the value specified at address P is assumed to be obtained using binary values. The following table lists the path numbers and corresponding binary values.

Path number	Binary value (decimal number)
1	1
2 or Loader path 1(1 control path)	2
Loader path 1 (2 control paths) or Loader path 2(1 control path)	4
Loader path 2 (2 control paths)	8

The bit position of each path in binary representation is shown below.



To make all of paths 1, 2, and 3 wait for one another, the P value is obtained as follows:

Binary value of path 1	1 (0000 0000 0000 0001)
Binary value of path 2	2 (0000 0000 0000 0010)
Binary value of path 3	4 (0000 0000 0000 0100)
Sum	7 (0000 0000 0000 0111)

All of the three paths can be made to wait for one another by specifying P7 together with an M code for waiting.

- Waiting specified with a combination of path numbers

When bit 1 (MWP) of parameter No. 8103 is set to 1, the value specified at address P is assumed to be a combination of path numbers. The following table lists the path numbers and corresponding values.

Path number	Value (decimal number)
1	1
2 or Loader path 1(1 control path)	2
Loader path 1 (2 control paths) or Loader path 2(1 control path)	3
Loader path 2 (2 control paths)	4

To make all of paths 1, 2, and 3 wait for one another, the P value is a number consisting of 1, 2, and 3.
Example) P123
There are no restrictions on the order in which the numeric characters are specified, the following six possible values can be specified:

P123, P132, P213, P231, P312, P321

Path numbers specified in combination in different orders for different paths are effective as long as the numbers of the relevant paths are specified.

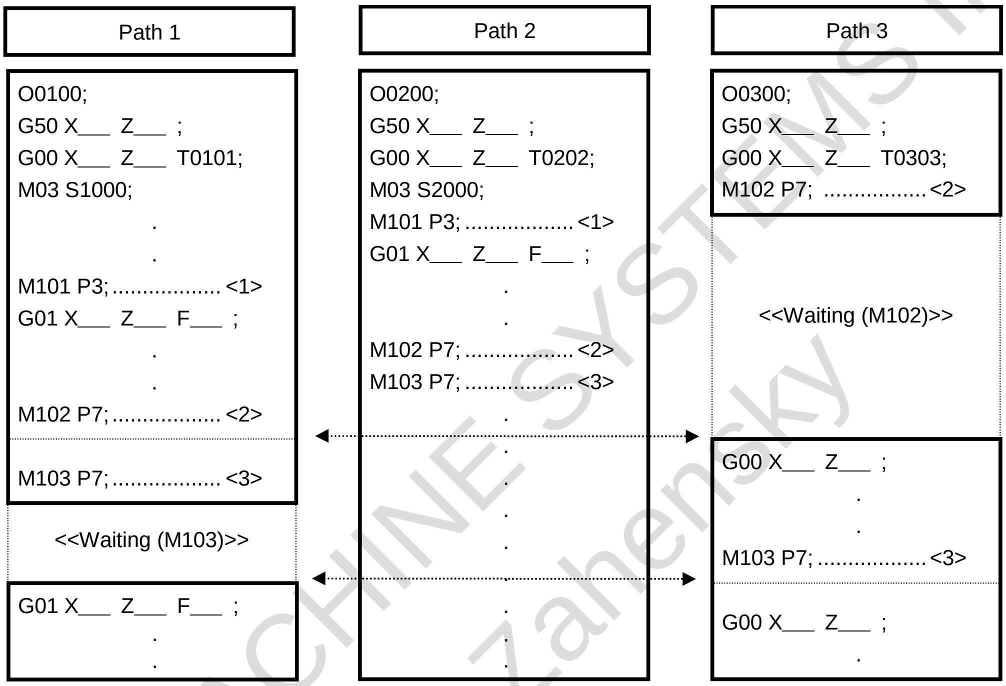
Example)

The following are treated as the same P value and these paths can be made to wait for one another:
M200P123 for path 1, M200P231 for path 2, and M200P321 for path 3

Program example

- When the value specified at P is obtained using binary values

Assume that the no-wait signal NOWT <G0063.1> for path 2 (NMWT<G1063.7> for a system with three or more paths) is set to "1" and M101 to M103 (parameter No. 8110 = 101 and parameter No. 8111 = 103) are set as M codes for waiting. In this case, programs O100, O200, and O300 for individual paths are executed as follows:



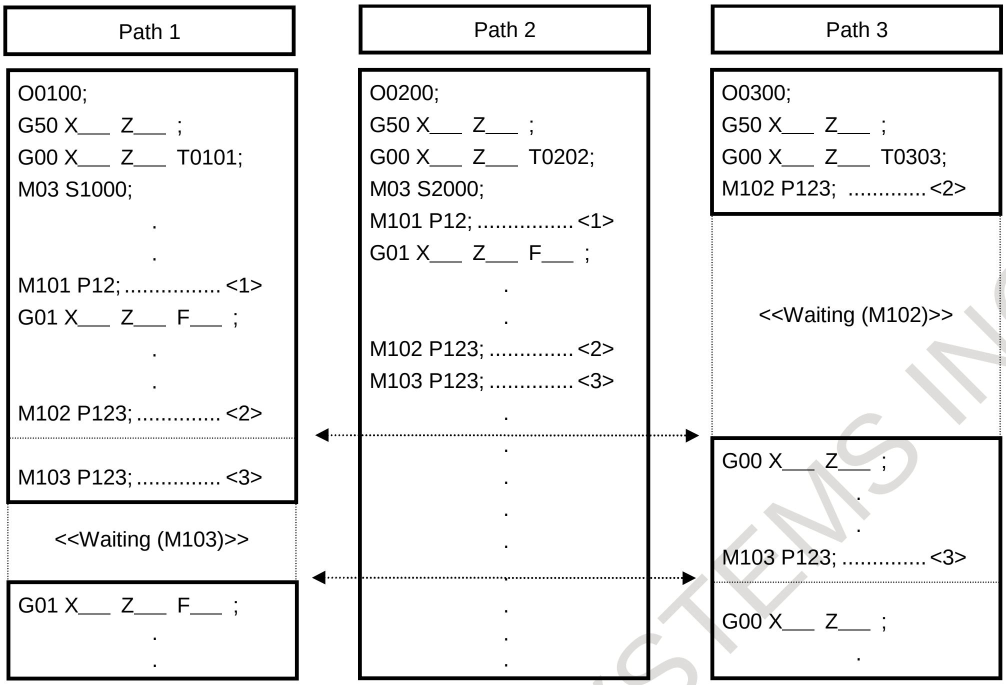
<1> M101 P3; (making paths 1 and 2 wait for each other)
When the no-wait signal for path 2 is set to "0", paths 1 and 2 wait for each other. Because the signal is set to "1", however, paths 1 and 2 ignore the M code for waiting and immediately execute the next block.

<2> M102 P7; (making paths 1, 2, and 3 wait for one another)
In this example, path 3 waits for processing on paths 1 and 2 to terminate. Because the no-wait signal for path 2 is set to "1", however, path 3 assumes that waiting is completed when processing on path 1 terminates, and executes the next block.

<3> M103 P7; (making paths 1, 2, and 3 wait for one another)
In this example, paths 1 and 2 wait for processing on path 3 to terminate. Because the no-wait signal for path 2 is set to "1", however, path 2 does not wait for processing on path 3 to terminate and executes the next block, but path 1 waits for path 3.

- When the value specified at P is obtained using path numbers in combination

Assume that the no-wait <G0063.1> signal for path 2 (NMWT<G1063.7> for a system with three or more paths) is set to "1" and M101 to M103 (parameter No. 8110 = 101 and parameter No. 8111 = 103) are set as M codes for waiting. In this case, programs O100, O200, and O300 for individual paths are executed as follows:



<1> M101 P12; (making paths 1 and 2 wait for each other)
When the no-wait signal for path 2 is set to "0", paths 1 and 2 wait for each other. Because the signal is set to "1", however, paths 1 and 2 ignore the M code for waiting and immediately execute the next block.

<2> M102 P123; (making paths 1, 2, and 3 wait for one another)
In this example, path 3 waits for processing on paths 1 and 2 to terminate. Because the no-wait signal for path 2 is set to "1", however, path 3 assumes that waiting is completed when processing on path 1 terminates, and executes the next block.

<3> M103 P123; (making paths 1, 2, and 3 wait for one another)
In this example, paths 1 and 2 wait for processing on path 3 to terminate. Because the no-wait signal for path 2 is set to "1", however, path 2 does not wait for processing on path 3 to terminate and executes the next block, but path 1 waits for path 3.

Signal

You can invalidate waiting by using the no-wait signal.
The waiting M code of a path for which the no-wait signal is "1" is ignored by other paths, thereby invalidating waiting. You use this signal if you are to perform automatic operation with a program containing a waiting M code, but you want to keep a certain path stopped while performing automatic operation in the other paths. For the path that you want to keep stopped, set the no-wait signal to "1".

No-wait signal

NOWT<G0063.1>(for path common signal interface)
NMWT<Gn063.7>(for path individual signal interface)

[Classification] Input signal

[Function] Specifies whether to synchronize the paths by the waiting M code.

[Operation] When this signal turns to "1" the paths are not synchronized by the M code. The M code for waiting specified in a machining program is ignored.
When this signal turns to "0", the paths are synchronized by the M code. When the M code for waiting is specified for one path, the CNC waits for the corresponding M code of another path to be issued, then starts executing the next block.

Waiting signal
WATO<F0063.6>(for path common signal interface)
WATO<Fn063.6>(for path individual signal interface)

[Classification] Output signal

[Function] Indicates that the M code is waiting for a path.

[Output cond.] These signals are "1" as long as:

- One path is waiting for another path. That is, the signal stays "1" for the period from when the M code for waiting is issued to one path to when the corresponding M code is issued to another path.

This signal is "0" as long as:

- Neither of the paths are waiting for the other.

> **NOTE**
> When the path common signal interface (bit 0 (MWT) of parameter No. 8103 = 1) is used, F0063.6=1 results even if path 2 is placed in the wait state.
> So, when the wait state needs to be checked with each path, use the path individual signal interface (bit 0 (MWT) of parameter No. 8103 = 0).

Signal address

- For path common signal interface

	#7	#6	#5	#4	#3	#2	#1	#0
G0063							NOWT	
F0063		WATO						

- For path individual signal interface

	#7	#6	#5	#4	#3	#2	#1	#0
Gn063	NMWT							
Fn063		WATO						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8103							MWP	MWT

[Input type] Parameter input

[Data type] Bit

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 MWT As the signal interface for the waiting M code:

0: The path individual signal interface is used.

1: The path common signal interface is used.

This parameter can be selected only when 2-path control is used.

#1 MWP To specify a P command for the waiting M code/balance cut:

0: A binary value is used as conventionally done.

1: A path number combination is used.

8110	**Waiting M code range (minimum value)**
8111	**Waiting M code range (maximum value)**

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 ,100to99999999

A range of M code values can be set by specifying a minimum waiting M code value (parameter No. 8110) and a maximum waiting M code value (parameter No. 8111).
(parameter No. 8110) ≤ (waiting M code) ≤ (parameter No. 8111)
Set 0 in these parameters when the waiting M code is not used.

Alarm and message

Number	Message	Description
PS0160	MISMATCH WAITING M-CODE	<1> When different M codes are specified for path 1 and path 2 as waiting M codes without a P command. <2> When the waiting M codes are not identical even though the P commands are identical <3> When the waiting M codes are identical and the P commands are not identical (This occurs when a P command is specified with binary value.) <4> When the number lists in the P commands contain a different number even though the waiting M codes are identical (This occurs when a P command is specified by combining path numbers.) <5> When a waiting M code without a P command (2-path waiting) and a waiting M code with a P command (3-or-more-path waiting) were specified at the same time <6> When a waiting M code without a P command was specified for 3 or more paths.
PS0161	ILLEGAL P OF WAITING M-CODE	<1> When address P is negative <2> When a P value inappropriate for the system configuration was specified <3> When a waiting M code without a P command (2-path waiting) was specified in the system having 3 or more paths.

An alarm occurs in all the paths in which an alarm-related waiting M code has been executed, bringing them into a single block stop and placing them in the automatic operation stop state.
At this time, the paths other than the alarm-related ones are stopped immediately, entering the automatic operation idle state.

Caution

⚠ **CAUTION**

1 Be sure to specify a waiting M code in an independent block.
2 Unlike other M codes, waiting M codes are not output to PMCs.
3 If you want to independently operate one path only, you need not delete that waiting M code. Using the no-wait signal (NOWT for a 2-path system and NMWT for a 3-path system), you can invalidate the waiting M code specified in the machining program. For details, refer to the manual issued by the machine tool builder.
4 To use a waiting M code in mode for multiple M commands in a single block, the M code must be specified as the first M code.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	WAITING FUNCTION FOR PATHS

8.3 WAITING M CODES OF HIGH-SPEED TYPE

Overview

Waiting M codes of high-speed type are useful function to reduce the cycle time of a program operation.

Specification

In order to use Waiting M codes of high-speed type, it is necessary to set the range of the M codes to parameters Nos.8114 and 8115.
The usage of these type M codes is similar to normal waiting M codes. For instance, the ways to set a path by address P and to use a no-wait signal or so on are the same.

Format

M m (P p) ;

m : Number of an M code for waiting
p : (1) In the binary value specification mode, specify the sum of the binary values corresponding to the numbers of paths which are to wait for one another.
(2) In the path number specification mode, specify the numbers of all paths that are to wait for one another in combination.

NOTE
1 Specify a waiting M code only in a block.
2 The irrelevant command, which is specified with a waiting M code in a block, is ignored.

Detail

Specifying the path by address P

The way to specify the path by address P is the same as normal waiting M codes.

Waiting signal

Waiting signal WATO<F0063.6> (for path common signal interface) and WATO<Fn063.6> (for path individual signal interface) are output like normal waiting M codes.

No-wait signal

No-wait signal NOWT<G0063.1> (for path common signal interface) and NMWT<Gn063.7> (for path individual signal interface) are effective like normal waiting M codes.

Waiting M codes of high-speed type invalid signal

Waiting M codes of high-speed type are treated as normal waiting M codes when waiting M codes of high-speed type invalid signal NHSW<G0579.6> is set to "1".

Limitation

Display of the currently executing program

Instead of the block of waiting M code, the block before waiting M code is displayed as the currently executing program during waiting of high-speed type.

Function code signal and strobe signal

As for waiting M code of high-speed type, the function code signal and strobe signal are not output.

Modal display

Waiting M codes of high-speed type are not displayed as the information of M code modal.

Condition to treat waiting M codes of high-speed type as normal waiting M codes

When one of the following conditions is satisfied, waiting M codes of high-speed type are treated as normal waiting M codes.

- Waiting M codes of high-speed type invalid signal is set to "1".
- The single-block operation is enabled.
- Waiting M codes of high-speed type are specified in the same block as the program number.
- In manual handle retrace function, the checking mode synchronizing an operation with a manual handle pulse is selected.
- Animation or path drawing of dynamic graphic display is operated.

NOTE

1 Waiting between waiting M codes of high-speed type and waiting M codes of high-speed type treated as normal waiting M codes cannot be executed when setting bit 7 (HMA) of parameter No.11279 to 0. In this case, alarm (PS0160) “MISMATCH WAITING M-CODE” occurs.
Example)
When either path 1 or path 2 is under the single block operation and waiting by "waiting M code of high-speed type" is executed between path 1 and path2, alarm (PS0160) occurs.

2 Waiting between waiting M codes of high-speed type and waiting M codes of high-speed type treated as normal waiting M codes can be executed when setting bit 7 (HMA) of parameter No.11279 to 1. And, waiting M codes of high-speed type can be specified during Relation of cutter compensation by setting bit 7 (HMA) of parameter No.11279 to 1. In this case, waiting M codes of high-speed type are not work at a high-speed, waiting M codes of high-speed type are treated as normal waiting M codes, and the processing time of waiting M codes of high-speed type is equal with normal waiting M codes.

Auxiliary function output in moving axis

Waiting is not executed even if waiting M code of high-speed type is specified, when Auxiliary function output in moving axis (G50.9) is given in the same block. In this case, M code is treated as a normal auxiliary function.

Manual handle retrace function

In Manual handle retrace function, a backward movement cannot be executed while the block of waiting M codes of high-speed type is executing. The backward movement ends when an operation enters the next block of waiting M code of high-speed type during the backward movement. And ‘RVED’ is displayed.
However, in the checking mode synchronizing with a manual handle pulse, the backward movement can be executed when waiting M code of high-speed type was executed once during forward movement, because the M code is treated as a normal M code in this case.

Retrace

The backward movement cannot be executed by Retrace function in the block of waiting M codes of high-speed type. The backward movement ends when the next block of waiting M code of high-speed type just starts executing during the backward movement. And ‘RVED’ is displayed.

Waiting function by specifying start point

Waiting M codes of high-speed type cannot act as a waiting function by specifying start point. Alarm (PS0160) occurs when such a command is specified. Use normal waiting M codes to use waiting function by specifying start point.

Relation of cutter compensation

Waiting M codes of high-speed type cannot be specified during the following cutter compensation functions. Alarm (PS0160) occurs when such a command is specified. Set the bit 7 (HMA) of parameter No.11279 to 1, or use normal waiting M codes during the following cutter compensation functions.
- Cutter compensation and tool nose radius compensation

> **NOTE**
>
> Waiting M codes of high-speed type cannot be specified just after the cancel command of cutter compensation, tool nose radius compensation. Moreover, waiting M codes of high-speed type cannot be specified while the vector of cutter compensation is being held.

Multiple repetitive canned cycle

Waiting M codes of high-speed type cannot be specified during Multiple repetitive canned cycle G70 to G73. Alarm (PS0160) occurs when such a command is specified. Use normal waiting M codes during Multiple repetitive canned cycle G70 to G73.

Chamfering and corner R, Optional chamfering and corner R and Direct drawing dimension programming

In the condition to treat waiting M codes of high-speed type as normal waiting M codes, alarm (PS0051) “MISSING MOVE AFTER CNR/CHF” occurs when waiting M codes of high-speed type are specified during Chamfering and corner R or Optional chamfering and corner R. And alarm (PS0312) “ILLEGAL COMMAND IN DIRECT DRAWING DIMENSIONS PROGRAMMING” occurs when waiting M codes of high-speed type are specified during Direct drawing dimension programming.

Quick program restart

When the program is interrupted by reset or etc. while executing waiting of high-speed type, the previous block of waiting M codes is regarded as the last restart point, instead of the block of waiting M codes.

Output to the PMC

Waiting M codes are not output to the PMC unlike the other M codes.

Occurring alarm while waiting

The state of cycle start lamp signal STL<Fn000.5> is as follows when an alarm occurs while waiting.

	Cycle start lamp signal STL<Fn000.5>	
	When alarms PS, SR or IO occurs.	**When other alarms occurs.**
Waiting M codes of normal	STL = 1	STL = 0
Waiting M codes of high-speed type	STL = 0	STL = 0

Block start interlock signal

Block start interlock signal *BSL<Gn008.3> is invalid for waiting M codes of high-speed type. However, block start interlock signal *BSL<Gn008.3> is valid in the condition to treat waiting M codes of high-speed type as normal waiting M codes.

Real-time custom macro

Execution timing of each real time macro statement is delayed compared with normal waiting M codes when real time macro command is executed during the waiting by waiting M codes of high-speed type.
Please use normal waiting M codes for the waiting which is executed with real time macro command at the same time.

Signal

Waiting M codes of high-speed type invalid signal NHSW<G0579.6>

[Classification] Input signal

[Function] Waiting M codes of high-speed type are invalidated.

[Operation] When this signal becomes "1":
M codes setting by parameters Nos.8114 and 8115 are treated as normal waiting M codes.
When this signal becomes "0":
M codes setting by parameters Nos.8114 and 8115 are treated as waiting M codes of high-speed type.

> **NOTE**
> This signal cannot be changed during operating a program.
> The state of signal at cycle start is kept.

No-wait signal
NOWT<G0063.1> (for path common signal interface)
NMWT<Gn063.7> (for path individual signal interface)

[Classification] Input signal

[Function] Specifies whether to synchronize the paths by the waiting M code or not.

[Operation] When this signal turns to "1", inter-path synchronization by the M code is not performed. The waiting M code specified in a machining program is ignored.
When this signal turns to "0", inter-paths synchronization by the M code is performed. When the waiting M code is specified in one path, the CNC waits for the corresponding M code of another path to be issued, then starts executing the next block.

Waiting signal
WATO<F0063.6> (for path common signal interface)
WATO<Fn063.6> (for path individual signal interface)

[Classification] Output signal

[Function] Indicates that the M code is waiting for another path.

[Output cond.] These signals are "1" as long as:

- A path is waiting for another path. That is, the signal stays "1" for the period from when the path issues the M code for waiting to when another path issues the corresponding M code.

This signal is "0" as long as:

- Neither of the paths is waiting for the other.

> **NOTE**
> 1 Waiting signal might not be output when waiting is released in a very short time.
> 2 Waiting signal might keep being output at 16ms or less, just after waiting has been released.

Parameter

8114	Top number of waiting M codes of high-speed type

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0, 100 to 99999999

Set the top number of waiting M codes of high-speed type.
Waiting M codes of high-speed type cannot be used when the setting value is 0 or out of range.

8115	Number of waiting M codes of high-speed type

[Input type] Parameter input
[Data type] word
[Valid data range] 0 to 32767

Set the number of waiting M codes of high-speed type.
Waiting M codes of high-speed type cannot be used when the setting value is 0 or out of range.

> **NOTE**
> Set the range of waiting M codes of high-speed type not to overlap with the range of other waiting M codes. The M codes are regarded as waiting M codes of high-speed type if overlapping.

	#7	#6	#5	#4	#3	#2	#1	#0
11279	HMA							

[Input type] Parameter input
[Data type] Bit

#7 HMA When waiting M codes of high-speed are specified during Relation of cutter compensation, :
0: alarm occurs.
1: alarm not occurs.

> **NOTE**
> 1 Processing time of waiting M codes of high-speed type during Relation of cutter compensation is equal with waiting M codes setting by parameter No.8110 and 8111 when setting this parameter to 1.
> 2 Waiting M codes of high-speed type during Relation of cutter compensation are treated with non-buffering and non-movement block when setting this parameter to 1. Please note enough about the function which changes operation by the non-movement block (For example, cutter compensation and tool nose radius compensation or etc.)

Alarm and message

Number	Message	Description
PS0051	MISSING MOVE AFTER CNR/CHF	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.
PS0160	MISMATCH WAITING M-CODE	1. When different M codes were specified in path 1 and path 2 as waiting M codes without a P command. 2. When the waiting M codes were not the same even though the P commands were the same. 3. When the waiting M codes were the same and the P commands were not the same (This case occurs when a P command was specified with binary value.) 4. When the number lists in the P commands contain a different number even though the waiting M codes were the same (This case occurs when a P command was specified by combining path numbers.) 5. When a waiting M code without a P command (2-path waiting) and a waiting M code with a P command (3-or-more-path waiting) were specified at the same time 6. When a waiting M code without a P command was specified for 3 or more paths. 7. When waiting was executed between waiting M codes of high-speed type and waiting M codes of high-speed type which are treated as normal waiting M codes. 8. When waiting M codes of high-speed type were specified as waiting function by specifying start point.
PS0161	ILLEGAL P OF WAITING M-CODE	1. When the value of address P was negative. 2. When a P value inappropriate for the system configuration was specified. 3. When a waiting M code without a P command (2-path waiting) was specified in the system having 3 or more paths.
PS0312	ILLEGAL COMMAND IN DIRECT DRAWING DIMENSIONS PROGRAMMING	Direct input of drawing dimensions was commanded in an invalid format. An attempt was made to specify an invalid G code during direct input of drawing dimensions. Two or more blocks not to be moved exist in consecutive commands that specify direct input of drawing dimensions. Although non-use of commas (,) (bit 4 of parameter No. 3405 = 1) was specified for direct input of drawing dimensions, a comma was specified.

8.4 INTERFERENCE CHECK FOR EACH PATH

T

Overview

When tool posts on individual paths machine the same workpiece simultaneously, the tool posts can approach each other very closely. If the tool posts interfere with each other due to a program error or any other setting error, a serious damage such as a tool or machine destruction can occur.
The function "interference check for each path" decelerates and stops the tool posts on individual paths before they actually make contact with each other if a command is issued and they will interfere with each other as the result.

Bit 7 (IFP) of parameter No. 8140 can be used to specify whether to make a two-path interference check or multipath interference check.

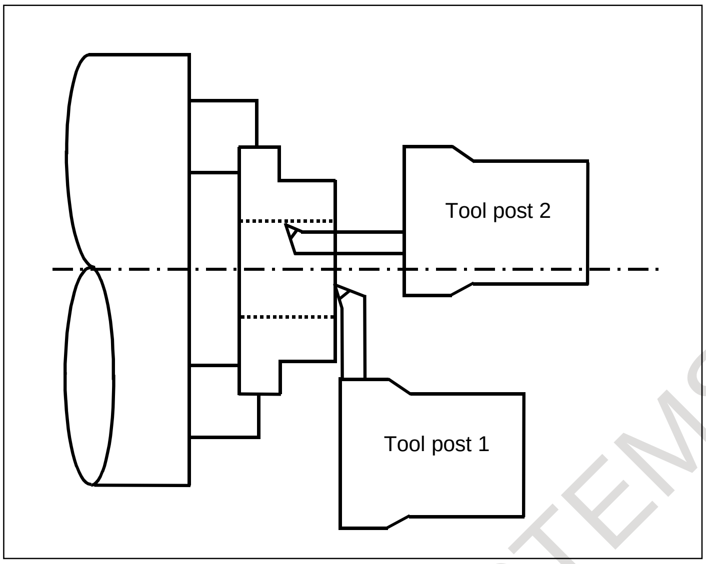


Fig. 8.4 (a)

The contours and shapes of the tool posts on individual paths are checked to determine whether or not interference occurs.

Explanation

- Data Setting for the Interference check for each path

To make a path interference check, data including the relationships between the tool posts on individual paths and interference forbidden areas (that is, tool shapes) needs to be set. The method of such data setting is described below.

- Setting the positions for reference points of tool posts on individual paths

X described below is the axis for which 1 (X-axis of the three basic axes) is set in parameter No. 1022 and Z is the axis for which 3 (Z-axis of the three basic axes) is set in the parameter. If an invalid value is set in this parameter, a path interference check is not made.

(1) Setting the reference points for a two-path interference check

When bit 7 (IFP) of parameter No. 8140 is set to 0, a two-path interference check is made.
When reference point return operation is completed with all axes (X1, Z1, X2, Z2), the reference point of tool post 1 is set at the origin of the ZX plane coordinate system. At this time, the position of the reference point of tool post 2 is set in a parameter.

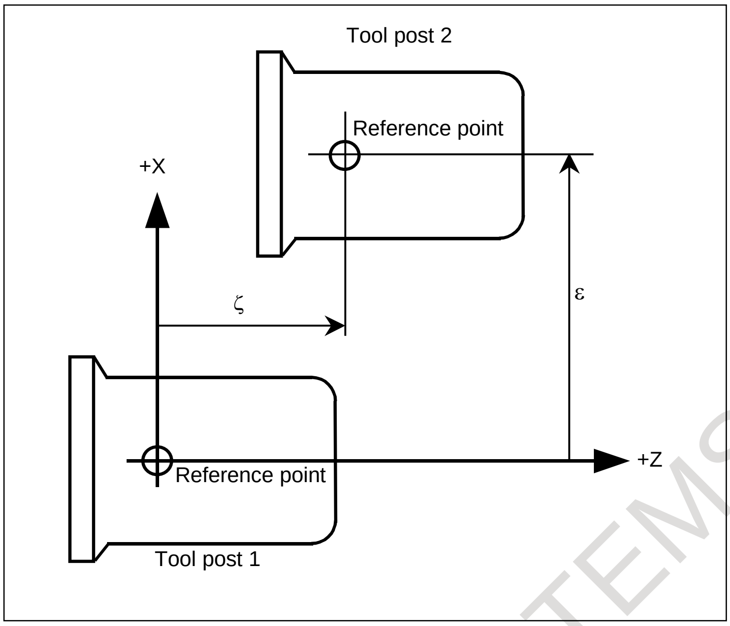


Fig. 8.4 (b)

In the ZX plane coordinate system at the origin of which the reference point of tool post 1 is set, set the X coordinate (ε) of the reference point of tool post 2 in parameter No. 8151, and its Z coordinate (ζ) in parameter No. 8152.
The unit of setting is the least command increment. For an axis subject to diameter specification, a diameter value is to be specified.
Measure (ε) and (ζ) when reference point return operation is completed with the four axes (X1, Z1, X2, Z2) (when the tools are in their reference points). When relative coordinate parameters Nos. 8151 and 8152 of the tool posts are to be updated, reference point return operation must always be completed with the four axes beforehand. Otherwise, the internally memorized relational positions of the tool posts are not updated to new parameter values.

- Setting the relationship between the coordinate systems for a two-path interference check

Set the relationship between the coordinate systems of the two tool posts using TY0 and TY1 (bits 0 and 1 of parameter No. 8140), with tool post 1 used as the reference.

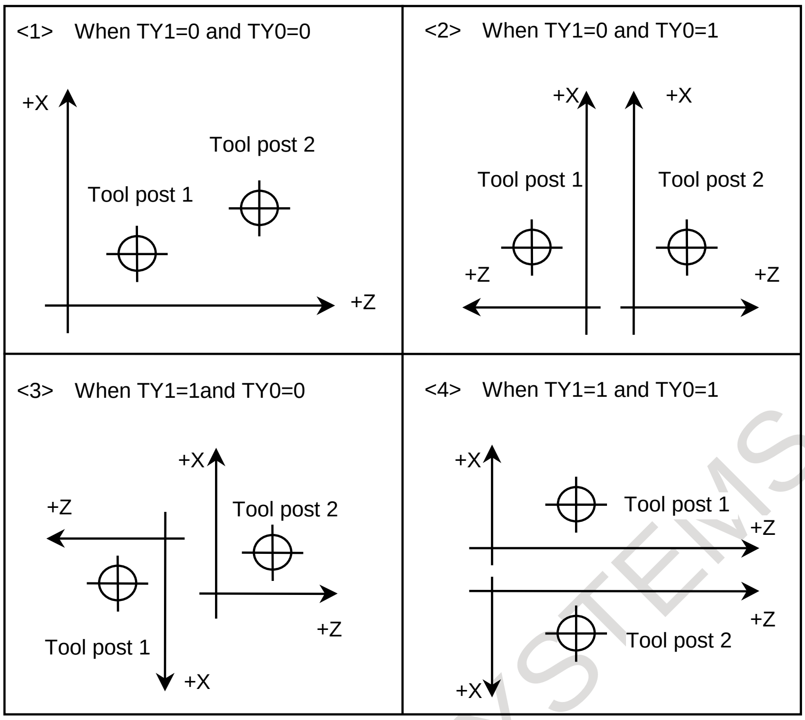


Fig. 8.4 (c)

(2) Setting the reference points for a multipath interference check

When bit 7 (IFP) of parameter No. 8140 is set to 1, multipath specifications are used, with which a multipath interference check is made. When reference position return operation is completed with all axes on all paths belonging to the same machine group, set the reference point of the tool post on path 1 belonging to the machine group as the origin of the ZX plane coordinate system. Also set the position of the reference point of the tool post on each of other paths in the coordinate system in parameters Nos. 8141 and 8143 (parameters for the X- and Z-axes are provided for each path).

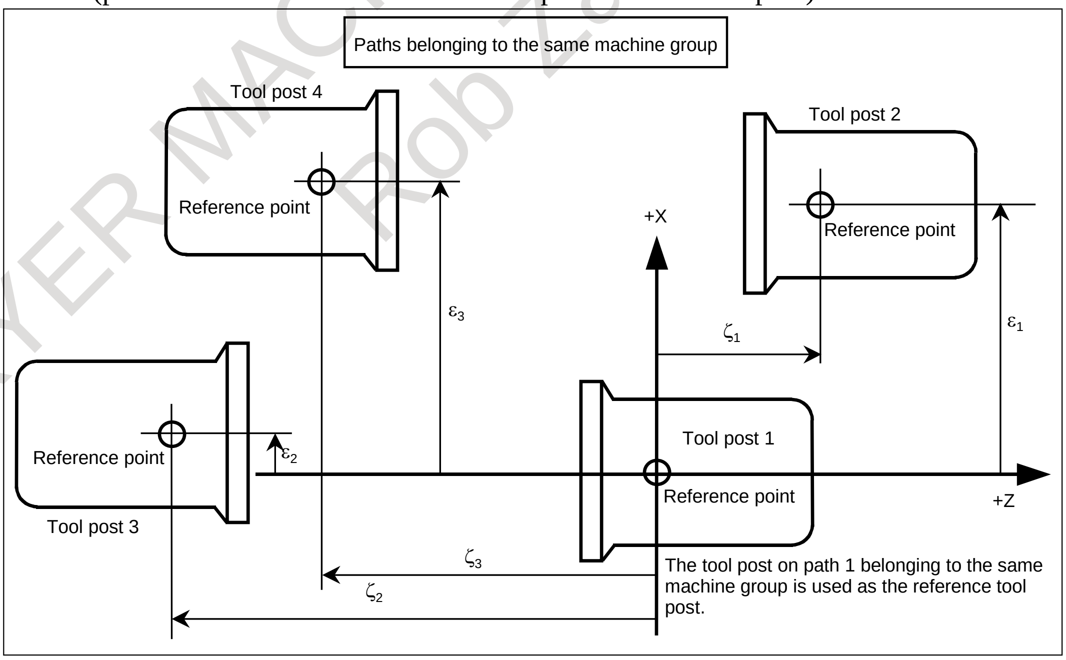


Fig. 8.4 (d)

In the example, Fig. 8.4 (d), there are the tool posts for four paths belonging to the same machine group. In the ZX plane coordinate system in which the reference point of tool post 1 on path 1 belonging to the same machine group is used as the origin, set the X coordinate (ε_1) of the reference point of tool post 2 on path 2 in parameter No. 8141 for path 2, and its Z coordinate (ζ_1) in parameter No. 8143.
In the same way, in the ZX plane coordinate system in which the reference point of tool post 1 is used as the origin, set the X coordinate (ε_2) of the reference point of tool post 3 on path 3 in parameter No. 8141 for path 3, and its Z coordinate (ζ_2) in parameter No. 8143. In the ZX plane coordinate system, set the X coordinate (ε_3) of the reference point of tool post 4 on path 4 in parameter No. 8141 for path 4, and its Z coordinate (ζ_3) in parameter No. 8143.
The unit of setting is the least input increment. For an axis subject to diameter specification, specify a diameter value.
Measure (ε_1, ζ_1), (ε_2, ζ_2), and (ε_3, ζ_3) when reference position return operation is completed with all axes (when the tools are in their reference positions). When updating a value of parameter No. 8141 or 8143 for each path, always perform reference position return operation with all axes on all paths. Otherwise, the internally stored relational positions of the tool posts are not updated to new parameter values.

- Setting the relationship between the coordinate systems for a multipath interference check

Select one of the following four relationships as the relationship between the coordinate system based on the reference point of the tool post on path 1 belonging to the same machine group and the position of each tool post and set it in parameter No. 8158 for the relevant path. The parameter setting indicates the positional relationship among the tool posts on the paths belonging to the same machine group.

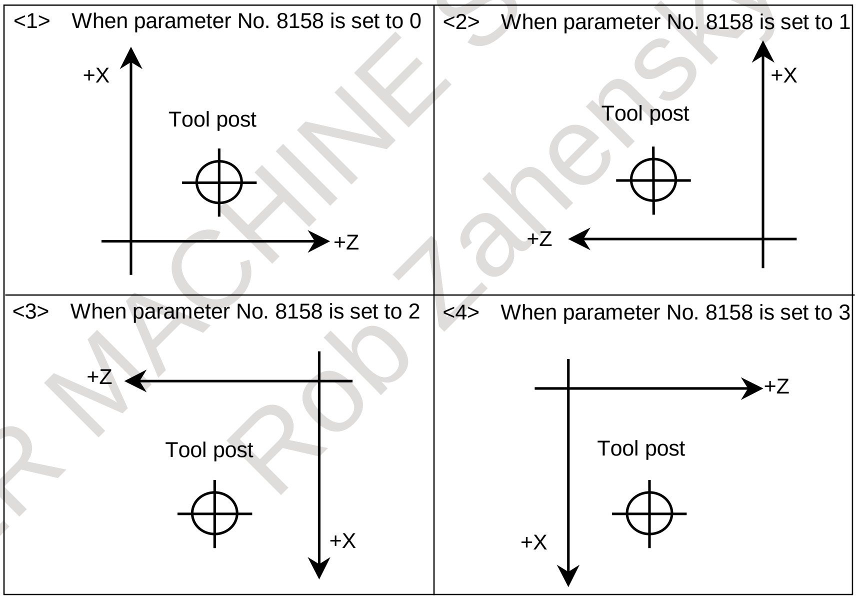


Fig. 8.4 (e)

For example, the parameter values are set as follows for each of the machine configurations in which the coordinate systems shown below are used.

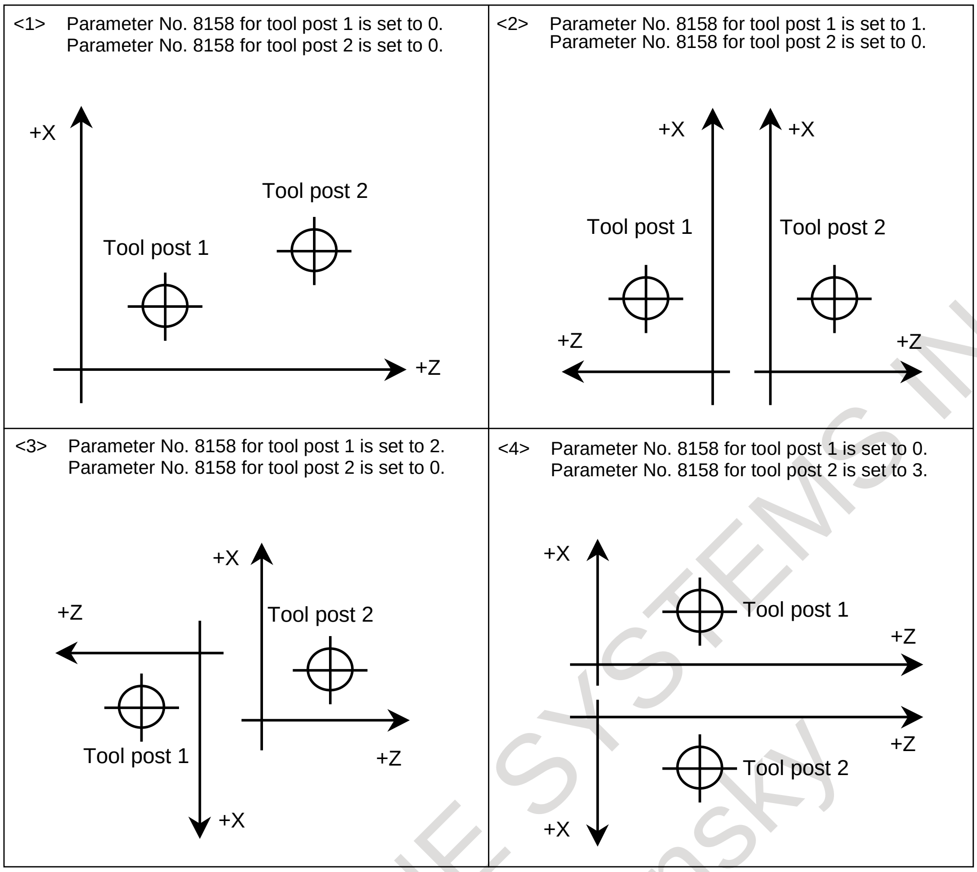


Fig. 8.4 (f)

- Specifying paths for which a multipath interference check is to be made

The following signals can be used to specify whether to make an interference check for the corresponding paths.

	#7	#6	#5	#4	#3	#2	#1	#0
Gn406					**ITF04**	**ITF03**	**ITF02**	**ITF01**

[Classification] Input signal

For these signals, the number assigned to each bit name indicates a path number.

These signals are available for individual paths (n = 0 to3, an area for signals for 4 paths is reserved). For each path, specify whether to make an interference check for each of other paths. To make an interference check, set the bit corresponding to a target path and the bit corresponding to the local path to 1.

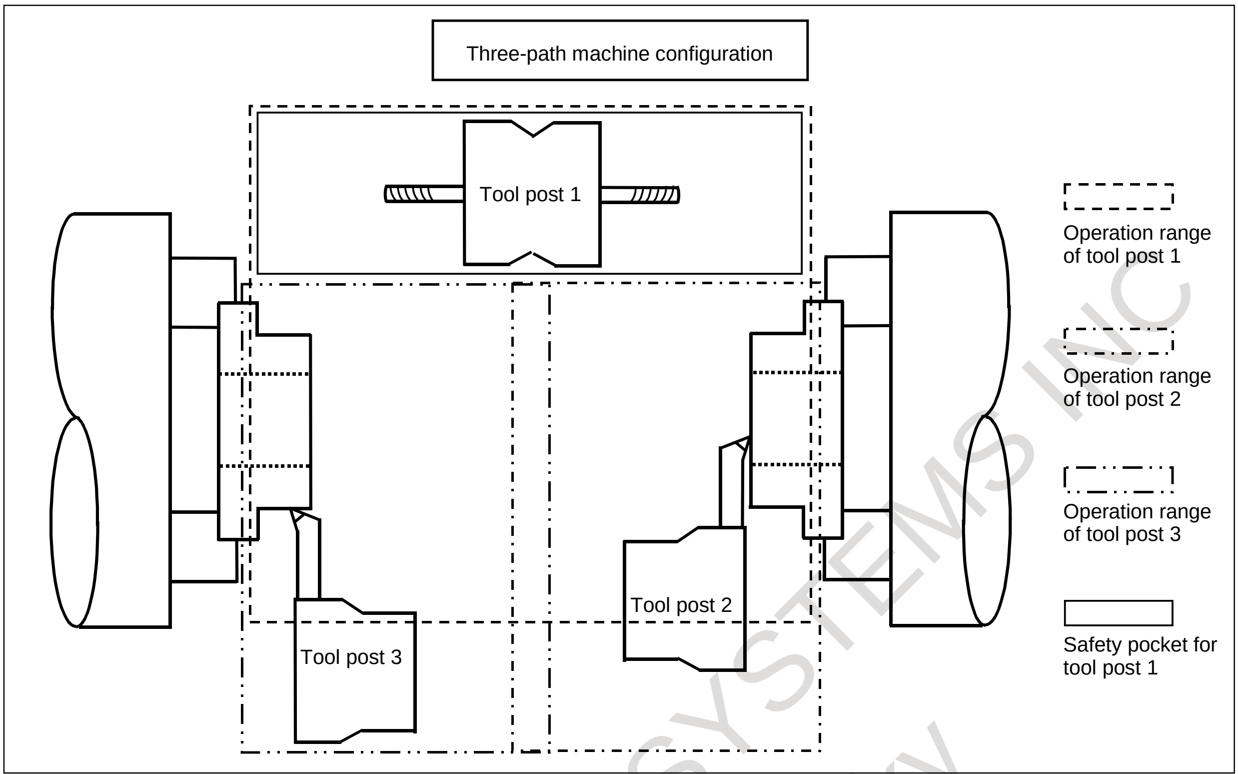


Fig. 8.4 (g)

For example, Fig. 8.4 (g), when the operation range of tool post 1 on path 1 overlaps with that of tool post 2 on path 2 and that of tool post 3 on path 3 as shown above, an interference check is required. In this sample machine configuration, however, tool post 1 is always positioned within the safety pocket in which the tool post does not interfere with path 2 or 3 when tool post 2 or 3 operates. In this case, for signals Gn406 for path 1, set the signals corresponding to paths 2 and 3 to 1 because an interference check must be made for the paths.

Path 1 Gn406 = 00000111

Paths 2 and 3 do not interfere with path 1, but the operation ranges of tool posts 2 and 3 overlap each other. In this case, an interference check is required and the setting is as follows:

Path 2 Gn406 = 00000110
Path 3 Gn406 = 00000110

- Setting the interference forbidden area (setting common to conventional and multipath specifications)

An interference forbidden area is set using a combination of two rectangular areas. The examples are shown below.

(Example 1)

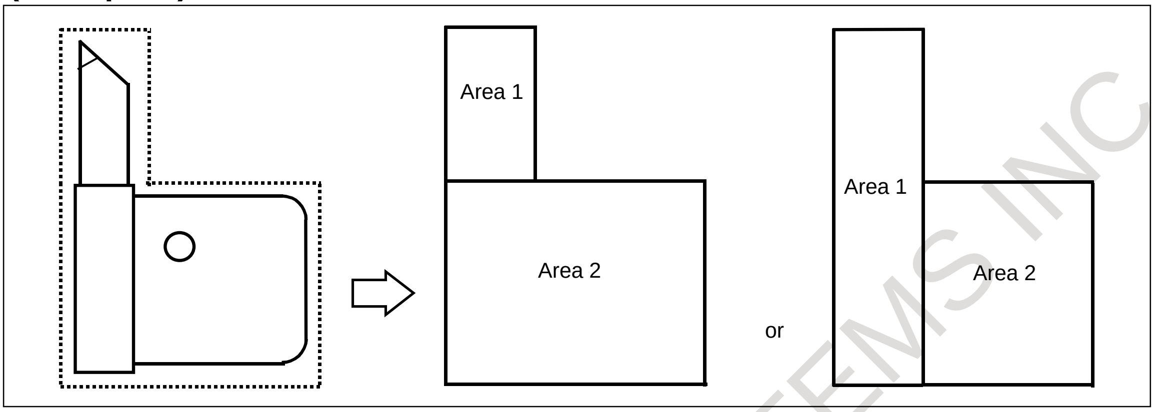


Fig. 8.4 (h)

(Example 2)

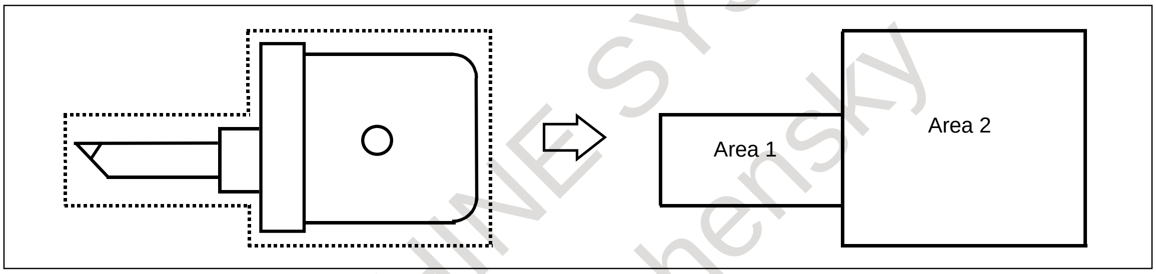


Fig. 8.4 (i)

The relative coordinates of the upper and lower ends (points A and B and points C and D shown below) of each of two rectangles are set, with the reference position of the tool post set as the origin.

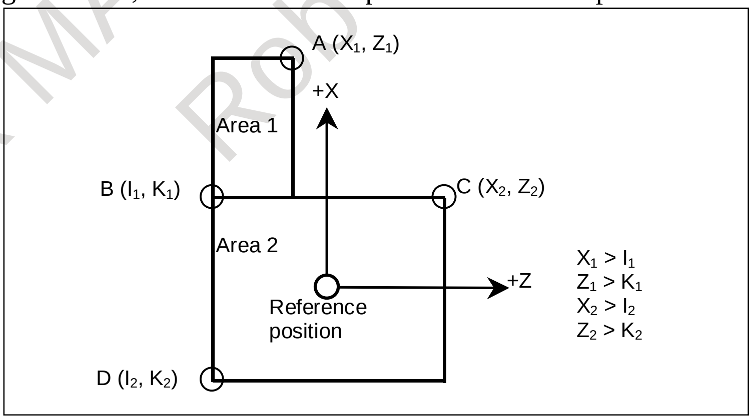
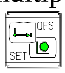


Fig. 8.4 (j)

Display and setting of an interference forbidden area in a multipath interference check

Display and set tool figure data (interference forbidden area) by performing the following operations:

(1) With the path selection signal, select a path for which an interference forbidden area is to be displayed and set in a multipath interference check.

(2) Press the function key .

(3) Press the chapter selection soft key [TOOL FIGURE].

(4) Display the screen including a tool number for which data is to be set.
Method 1: Switch the screen display by pressing the page key or cursor key.
Method 2: Enter a tool number to be set then press the [NO.SRH] soft key.

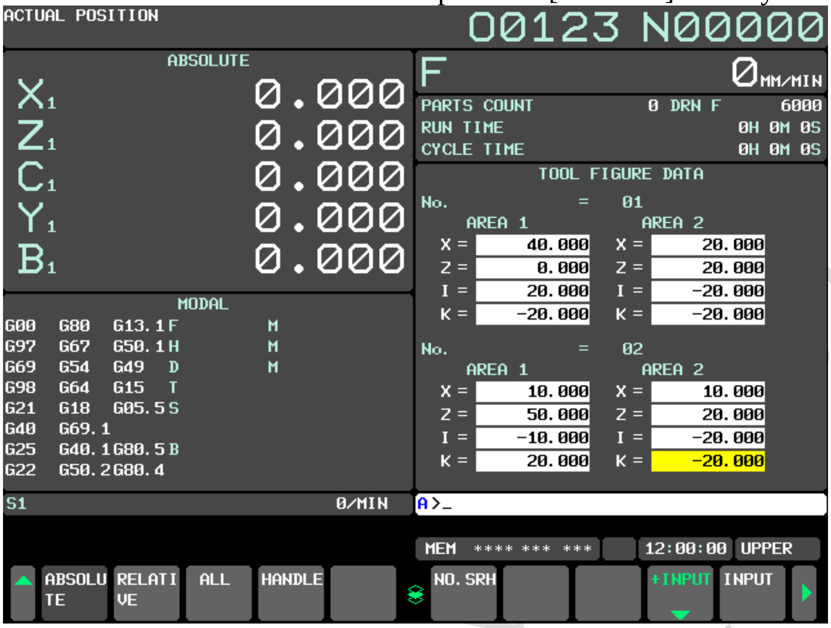


Fig. 8.4 (k)

(5) With the cursor keys, move the cursor to data to be set.
(When setting data for point A, move the cursor to X and Z. When setting data for point B, move the cursor to I and K.)

(6) Enter the coordinate values of point A or point B by using numeric keys.
(The decimal point can also be entered.)
Enter the coordinate values of point A or point B by using numeric keys.
(The decimal point can also be entered.)

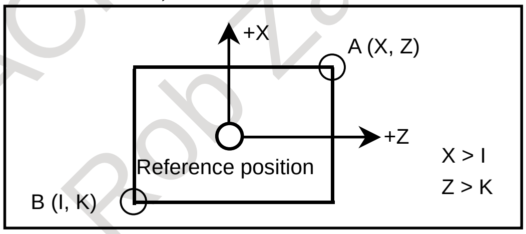


Fig. 8.4 (l)

(7) By pressing the soft key [INPUT], the entered coordinate values are set.
(To add an entered value to the already set data, press the soft key [+INPUT].)

NOTE

Tool number

Tool figure data needs to be set for each tool number. A tool number used here means an offset number. When both of geometry offset and wear offset are used, a tool number corresponds to a wear offset number. When multiple offset numbers are used for the same tool, the same data needs to be set as tool figure data.

- Conditions for Making a Path Interference Check

A path interference check is made when conditions listed below are satisfied.

(1) Common conditions

- Bit 4 (IFE) of parameter No. 8140 for enabling the path interference check function is set to 0.
- The X- and Z-axes of the three basic axes are set for parameter No. 1022 for individual paths for which a path interference check is to be made.
- After power is turned on, reference point return operation is completed with all axes. (When an absolute-position detector is used, the matching between a machine position and absolute-position detector position must be completed.)
- Offset numbers other than 0 are specified using T codes for tool posts on the paths for which a path interference check is made.
- Values are input in either one or both of the areas of the tool shape data corresponding to the specified tool number and the relationships among the values are X > I and Z > K.
- When manual mode is used, bit 3 (IFM) of parameter No. 8140 for enabling the interference check for each path function in manual mode is set to 1.

(2) Conditions for a multipath interference check

A multipath interference check is made when the conditions below as well as the common conditions are satisfied:

- Bit 7 (IFP) of parameter No. 8140 for enabling the multipath interference check function is set to 1.
- The path associated with each signal of Gn406 for each path exists.

(3) When the interference check conditions are satisfied

When all conditions for making a path interference check are satisfied, the path-interference-check-in-progress signal is output to the PMC.

> ⚠ **WARNING**
> To make a path interference check, the tool number for which tool shape data is input must agree with the programmed tool number. An interference check cannot be made correctly if the tool is selected by manual operation or if no tool select command is specified at power-on.

Signal

- For a check for interference between two paths

Path interference check in progress signal TICHK<F0064.6>

[Classification] Output signal

[Function] Informs whether a path interference check is in progress.

[Output cond.] This signal goes to "1" when:

- All conditions necessary to perform a path interference check on the tool posts of the first and second paths have been satisfied.

This signal goes to "0" when:

- Any one of the conditions necessary to perform a path interference check on the tool posts of the first and second paths have not been satisfied.

Path interference alarm signal TIALM<F0064.7>

[Classification] Output signal

[Function] Informs whether an alarm has occurred due to a path interference check.

[Output cond.] This signal goes to "1" when:

- The two tool posts are judged to interfere with each other while a path interference check is in progress on the tool posts of the first and second paths.

This signal goes to "0" when:

- The two tool posts are judged not to interfere with each other while a path interference check is in progress on the tool posts of the first and second paths.
- A path interference check is not performed (the path interference check signal TICHK is "0")

- For a multi-path interference check

Path interference check association signal ITF01 to ITF04 <Gn406>

[Classification] Input signal

[Function] Specifies the path subject to a path interference check.

ITFx

x : 01 Performs a path interference check with the tool post of the first path.
02 Performs a path interference check with the tool post of the second path.
03 Performs a path interference check with the tool post of the third path.
04 Performs a path interference check with the tool post of the fourth path.

[Operation] Performs a path interference check with the path for which "1" is set.
Be sure to set the bit corresponding to the local path to "1".

Path interference check in progress signal TICHK<Fn064.6>

[Classification] Output signal

[Function] Notifies whether a path interference check is in progress.

[Output cond.] This signal goes to "1" when:

- At least one of the paths associated by means of the Gn406 signals enters a state in which it can be subject to a path interference check with the tool post of the local path.

  Example)

  Assuming that the setting of the Gn406 signal of the 1st path is 00001101, and other settings are such that an interference check is to be performed with the tool posts of the third and fourth paths, F0064.6 of the first path will become "1" when the first path and either the third or fifth path enter the state in which they can be subject to a path interference check.

  If, at this time, the setting of the Gn406 signal of the third path is 00001100, and other settings are such that an interference check is to be performed with the tool post of the fourth path only, F2064.6 of the third path will become "1" when the third and fifth path enter the state in which they can be subject to a path interference check.

  As described above, ON/OFF timing differs depending on the settings of the Gn406 signals of each path.

This signal goes to "0" when:

- At least one of the conditions necessary to perform a path interference check on the too posts of the path associated by means of the Gn406 and Gn407 signals and of the local path is not satisfied.

Path interference alarm signal TIALM <Fn064.7>

[Classification] Output signal

[Function] Notifies whether an alarm has occurred due to a check for path interference.

[Output cond.] This signal goes to "1" when:

- The signal of the local path, Fn064.7 goes "1" if, while a path interference check is in progress on the tool posts of the paths associated by means of the Gn406 signals and of the local path, at least one of the associated paths is judged to have interfered because the tool post of the local path moved.

This signal goes to "0" when:

- While a path interference check is in progress on the tool posts of the paths associated by means of the Gn406 signals and of the local path, none of the associated paths is judged to have interfered.
- A path interference check is not in progress (the path interference check signal TICHK is "0")

Signal address

- For a 2-path interference check

	#7	#6	#5	#4	#3	#2	#1	#0
F0064	TIALM	TICHK						

- For a multi-path interference check

	#7	#6	#5	#4	#3	#2	#1	#0
Gn406					ITF04	ITF03	ITF02	ITF01

	#7	#6	#5	#4	#3	#2	#1	#0
Fn064	TIALM	TICHK						

Caution

- Items common to both 2-path interference and multi-path interference checks

NOTE

1 If an alarm occurs due to a path interference check, first switch to manual mode, manually retract the tool post of the path in which the alarm occurred to a position without interference, then perform a reset to cancel the alarm on the control unit. By manually retracting the tool post, the TIALM signal <G0064.7> will become "0" at the point when the tool post associated with the interference is judged not to interfere. (This can be used as a guideline: If, after a tool post has interfered, retracting it manually, you have only to retract it until the TIALM signal <G0064.7> becomes "0".)

2 If an alarm occurs due to a path interference check, the axis and direction of movement that were assumed at that time will be stored, so that the tool can no longer be moved in the stored direction along the stored axis until the alarm on the control unit is canceled with a reset. (Any start that will result in further interference will be prohibited.)

3 If an interference alarm occurs, the tool post will stop before the interference position if bit 7 (BFA) of parameter No. 1300 is set to 1.
If this parameter is set to 0, the CNC and the machine system will stop somewhat later than the correct time, the positions at which they actually stop will be closer to the other tool post that the interference position specified in tool figure data. For safety, therefore, the tool figure data must be somewhat larger than necessary. The excess distance L may be calculated from the rapid traverse rate.
L = (Rapid traverse rate) x 1/7500
If, for example, the rapid traverse rate is 15 m/min, the figure data must be larger by about 2 mm.

- Item for a 2-path interference check only

NOTE

Besides the common items, there is a note on a 2-path interference check only. If, during a path interference check, the tool posts of the two paths are judged to have interfered with each other, both the paths are decelerated to a stop and the control unit enters the alarm state. If this occurs, the path interference alarm signal TIALM <F0064.7> is set to "1" to notify that an alarm has occurred in a path interference check.

- Item for a multi-path interference check only

> **NOTE**
> Besides the common items, there is a note on a multi-path interference check only.
> If, during a path interference check, the tool posts of the paths associated with the Gn406 signals of each path are judged to have interfered with each other, all the paths associated are decelerated to a stop, and the paths related to the interference (paths in which movement was in progress) enter the alarm state. Other associated paths enter the operation pause state. If this occurs, the tool post interference alarm signal TIALM <Fn064.7> of the path related to the interference is set to "1" to notify that a check has occurred in a path interference check.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA							

[Input type] Setting input
[Data type] Bit path

#7 BFA When the stored stroke check 1, 2, or 3 alarm is issued, an interference alarm is issued with the inter-path interference check function (lathe system), or a chuck/tail stock barrier (lathe system) alarm is issued:
0: The tool stops after entering the prohibited area.
1: The tool stops before the prohibited area.

> **NOTE**
> 1. This parameter is valid even in AI contour control.
> 2. This parameter is invalid for the slave axis under axis synchronous control.

	#7	#6	#5	#4	#3	#2	#1	#0
8140	IPF		ZCL	IFE	IFM	IT0	TY1	TYO

[Input type] Parameter input
[Data type] Bit

#0 TY0
#1 TY1 This parameter sets the coordinate system relationship between two tool posts based on the tool post of path 1.
This parameter is used for checking the interference between two paths when bit 7 (IPF) of parameter No. 8140 is set to 0.

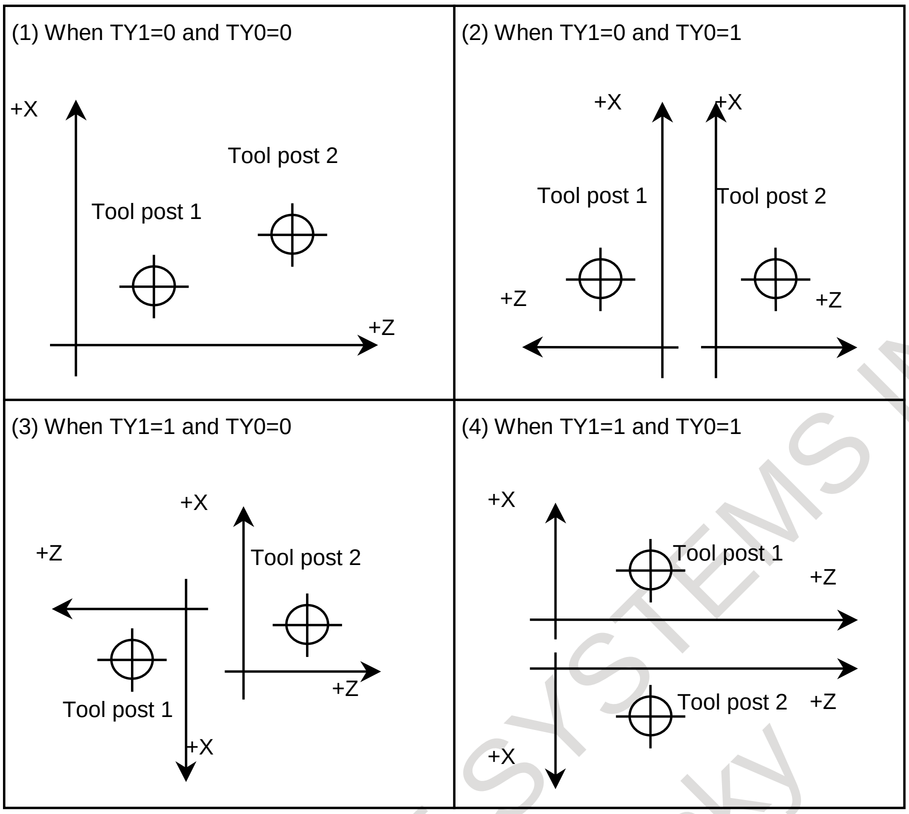


Fig. 8.4 (m)

#2 IT0 When offset number 0 is specified by the T code,
- 0: Checking interference between paths is stopped until an offset number other than 0 is specified by the next T code.
- 1: Checking interference between paths is continued according to the previously specified offset number.

#3 IFM In manual mode, a interference check between paths is:
- 0: Not performed.
- 1: Performed.

#4 IFE Interference check between paths is:
- 0: Performed.
- 1: Not performed.

#5 ZCL Specifies whether interference along the Z axis is checked while checking interference between paths.
- 0: Checked.
- 1: Not checked (Only interference along the X axis is checked.)

#7 IPF In inter-path interference checking:
- 0: The interference between two paths is checked.
- 1: The interference among multiple paths is checked.

Even in two-path control, the specification of a multi-path interference check can be applied.

If this parameter is set to 0 when three or more paths are controlled, a two-path interference check is made only with path 1 and path 2.

8141	Distance along the X axis between the reference positions of tool post 1 and tool post n in the same machine group

8143	Distance along the Z axis between the reference positions of tool post 1 and tool post n in the same machine group

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Each of these parameters sets the distance between the reference positions of the tool post of path 1 and tool post of each path in the same machine group.
Set 0 in parameters Nos. 8141 and 8143 for tool post 1 of each machine group.
For the lathe system, only a Z-X coordinate system based on parameters Nos. 8141 and 8143 is used for setting.

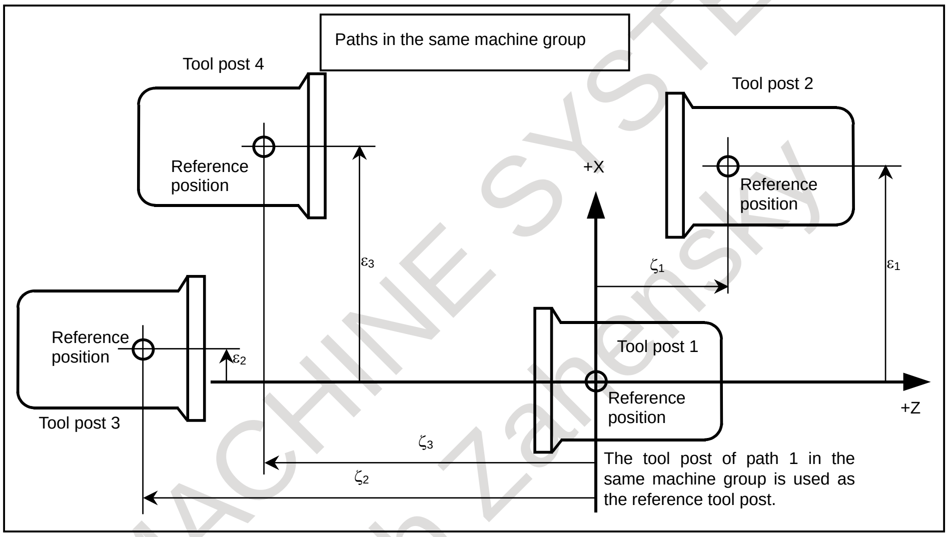


Fig. 8.4 (n)

In the example above (Fig. 8.4 (n), the same machine group contains tool posts for four paths. In the ZX plane coordinate system with its origin placed at the reference position of tool post 1 of path 1 in the same machine group, the position of the reference position of tool post 2 of path 2 is specified by setting the value ε_1 of the X component in parameter No. 8141 for path 2 and by setting the value ζ_1 of the Z component in parameter No. 8143 for path 2.
Similarly, In the ZX plane coordinate system with its origin placed at the reference position of tool post 1, the position of the reference position of tool post 3 of path 3 is specified by setting the value ε_2 of the X component in parameter No. 8141 for path 3 and by setting the value ζ_2 of the Z component in parameter No. 8143 for path 3. In the ZX plane coordinate system with its origin placed at the reference position of tool post 1, the position of the reference position of tool post 4 of path 4 is specified by setting the value ε_3 of the X component in parameter No. 8141 for path 4 and by setting the value ζ_3 of the Z component in parameter No. 8143 for path 4.
The unit of setting is the least input increment. For an axis based on diameter specification, make a setting using a diameter value.

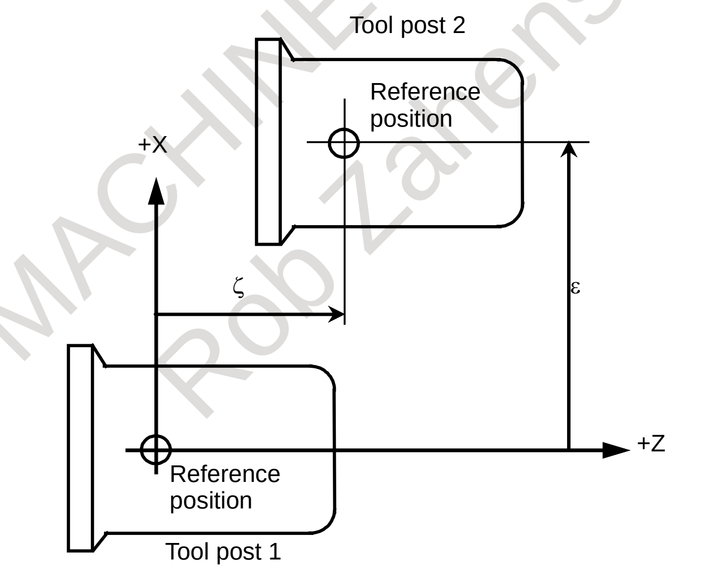
> ⚠ **WARNING**
> Measure (ε_1, ζ_1), (ε_2, ζ_2), and (ε_3, ζ_3) in the state where reference position return operation is completed for all axes (the tool is at the reference position.)
> After modifying parameters Nos. 8141 and 8143 for each path, be sure to perform a reference position return operation along all axes in all paths. Otherwise, the internally stored positional relationships of the tool posts are not updated to the newly set parameter values.

8151	**Distance along the X axis between the reference positions of tool posts 1 and 2**

8152	**Distance along the Z axis between the reference positions of tool posts 1 and 2**

[Input type] Parameter input
[Data type] Real
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

These parameters are used for checking the interference between two paths when bit 7 (IPF) of parameter No. 8140 is set to 0.
Each of these parameters sets the distance between the tool posts of two paths.



In the Z-X plane coordinate system with its origin at the reference position of tool post 1, set the X component value ε of the reference position of tool post 2 in parameter No. 8151 and set the Z component value ζ in parameter No. 8152.

Fig. 8.4 (o)

> ⚠ **WARNING**
> After modifying the parameter values, perform a manual reference position return operation for both tool posts. Otherwise, the internally stored positional relationships of the two tool posts are not updated to the newly set parameter values.

8158	Coordinate system pattern with the reference position based on the tool post of path 1 in the same machine group

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

This parameter is used for checking the interference among multiple paths when bit 7 (IPF) of parameter No. 8140 is set to 1.
This parameter sets a coordinate system pattern with the reference position based on the tool post of path 1 in the same machine group.

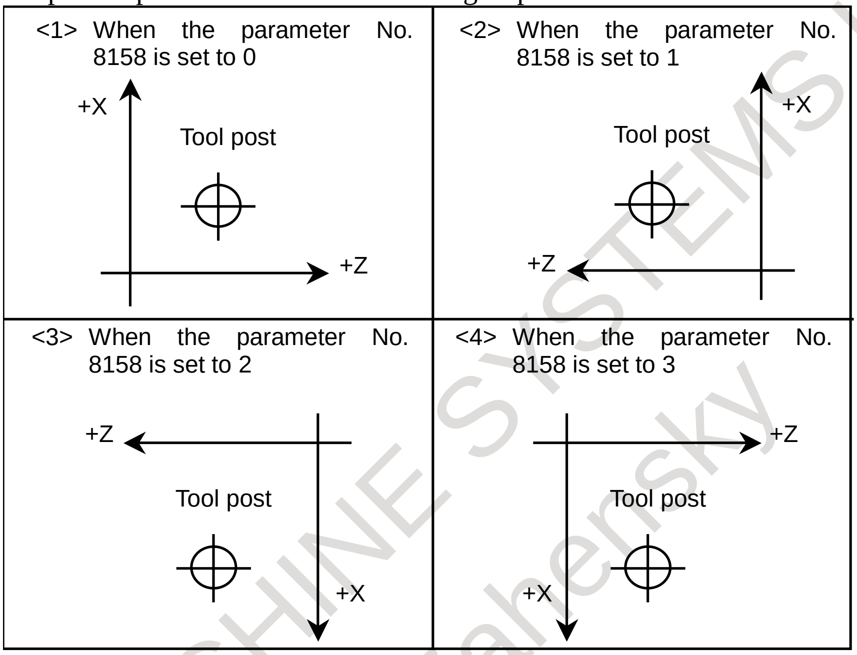


Fig. 8.4 (p)

Alarm and message

Number	Message	Description
PS0169	ILLEGAL TOOL GEOMETRY DATA	Incorrect tool figure data in interference check. Set correct data, or select correct tool figure data.
OT0508	INTERFERENCE:+	A tool moving in the positive direction along the n axis has fouled another tool post.
OT0509	INTERFERENCE:-	A tool moving in the negative direction along the n axis has fouled another tool post.

Alarms will occur individually for the axes associated with interference.

WARNING

⚠ WARNING

After setting parameters for the interference check function, tool figure data (contact prohibited area), etc., be sure to enter manual mode (parameter settings must be such that an interference check is enabled even in manual mode), let the tool post of each path interfere in all directions and confirm that a normal interference area has been set.

8.5 BALANCE CUTTING

T

Overview

When a thin workpiece is to be machined as shown below, a precise machining can be achieved by machining each side of the workpiece with a tool simultaneously; this function can prevent the warpage of the workpiece that can result when only one side is machined at a time (see the Fig. 8.5). When both sides are machined at the same time, the movement of one tool must be in phase with that of the other tool. Otherwise, the workpiece can vibrate, resulting in poor machining. With this function, the movement of one tool post can be easily synchronized with that of the other tool post.

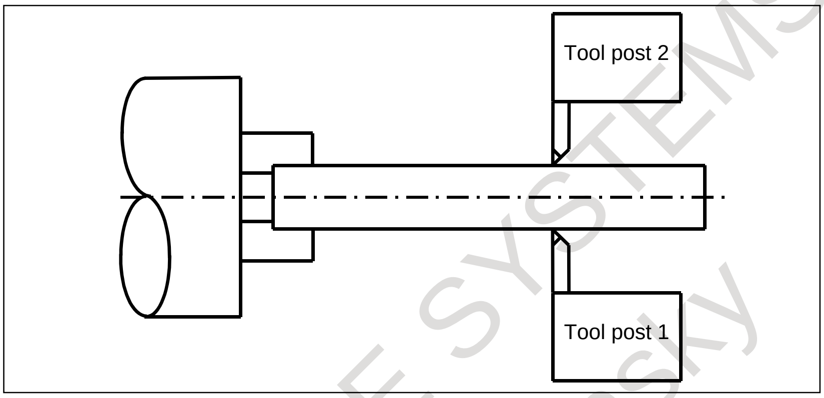


Fig. 8.5

Specifying G68, which turns balance cutting mode on, causes balance cutting to be performed with the tool post of path 1 and that of path 2. Alternatively, specifying address P in the same block as that containing the G64 command causes balance cutting to be performed between any tool posts.
Address P to be specified in the same block as G68, which turns balance cutting mode on can be set in either of two ways:

- When the bit 1 (MWP) of parameter No. 8103 is set to 0 :
  Balance cutting with a binary value specification
- When the bit 1 (MWP) of parameter No. 8103 is set to 1 :
  Balance cutting with a path number combination specification

NOTE
1 Balance cutting is not performed in dry run or machine lock state. G68 or G69 specified for one tool post is synchronized with G68 or G69 specified for the other tool post, however.
2 In the balance cut mode, G68 specified for one tool post is not synchronized with G68 specified for the other tool post. In the balance cut cancel mode, G69 specified for one tool post is not synchronized with G69 specified for the other tool post.
3 Balance cutting is not performed in a block in which 0 is specified for the travel distance.
4 Balance cutting is not performed when rapid traverse is specified.

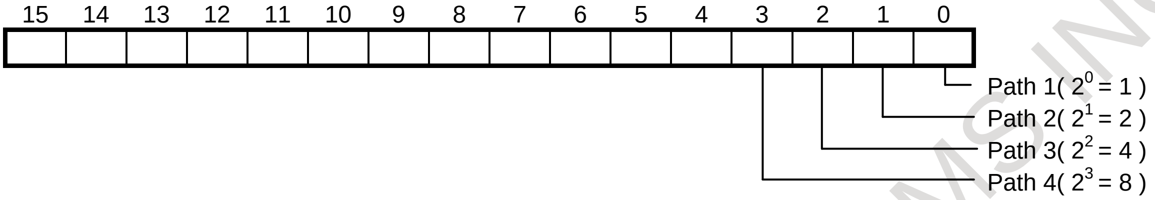
- Balance cut specified with binary values

When bit 1 (MWP) of parameter No. 8103 is set to 0, the value specified at address P is assumed to be obtained using binary values. The Table 8.5 (a) lists the path numbers and corresponding binary values.

Table 8.5 (a)

Path number	Binary value (decimal number)	Path number	Binary value (decimal number)
1	1	6	32
2	2	7	64
3	4	8	128
4	8	9	256
5	16	10	512

The bit position of each path in binary representation is shown below.



To perform balance cutting for all of paths 1, 2, and 3, the P value is obtained as follows:

Binary value of path 1	1 (0000 0000 0000 0001)
Binary value of path 2	2 (0000 0000 0000 0010)
Binary value of path 3	4 (0000 0000 0000 0100)
Sum	7 (0000 0000 0000 0111)

Balance cutting can be performed for all of the five paths by specifying P341 together with the balance cut G code (G68).

- Balance cut specified with a combination of path numbers

When bit 1 (MWP) of parameter No. 8103 is set to 1, the value specified at address P is assumed to be a combination of path numbers. The Table 8.5 (b) lists the path numbers and corresponding values.

Table 8.5 (b)

Path number	Value (decimal number)	Path number	Value (decimal number)
1	1	6	6
2	2	7	7
3	3	8	8
4	4	9	9
5	5	10	0

To perform balance cutting for all of paths 1, 2, and 3, the P value is a number consisting of 1, 2, and 3.
Example) P123
There are no restrictions on the order in which the numeric characters are specified, the following six possible values can be specified:

P123, P132, P213, P231, P312, P321

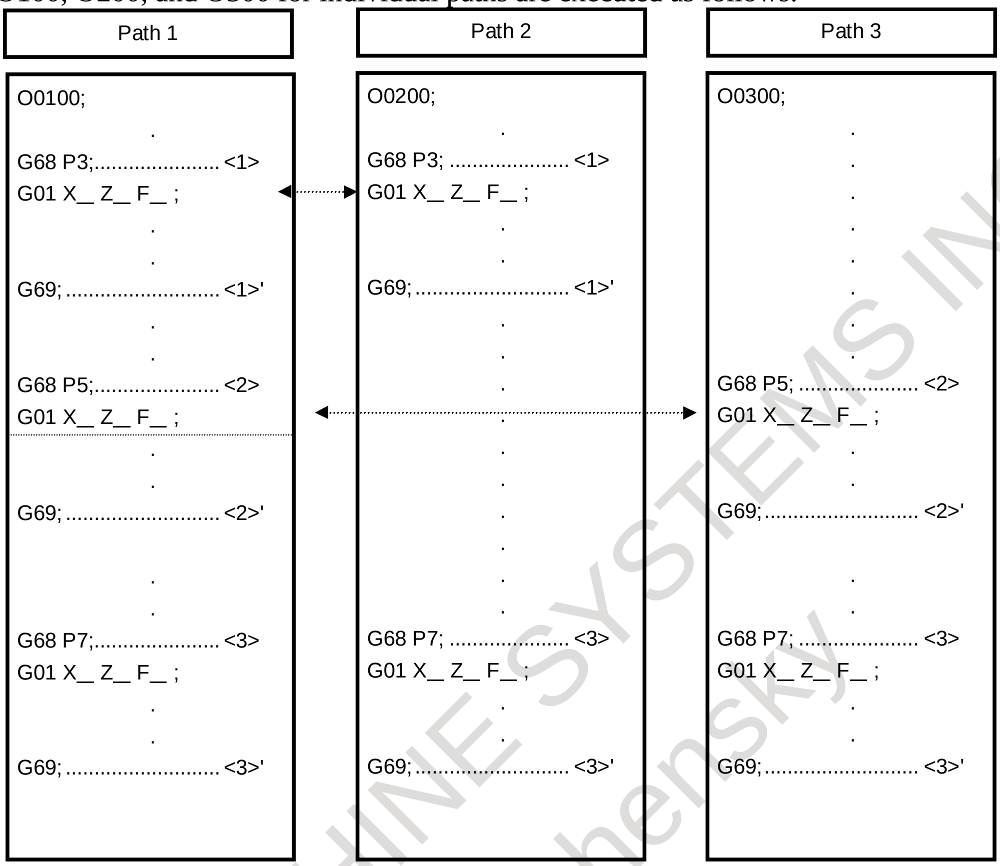
Path numbers specified in combination in different orders for different paths are effective as long as the numbers of the relevant paths are specified.
Example)

The following are treated as the same P value
M200P123 for path 1, M200P231 for path 2, and M200P321 for path 3

Program example

- When the value specified at P is obtained using binary values

Programs O100, O200, and O300 for individual paths are executed as follows:



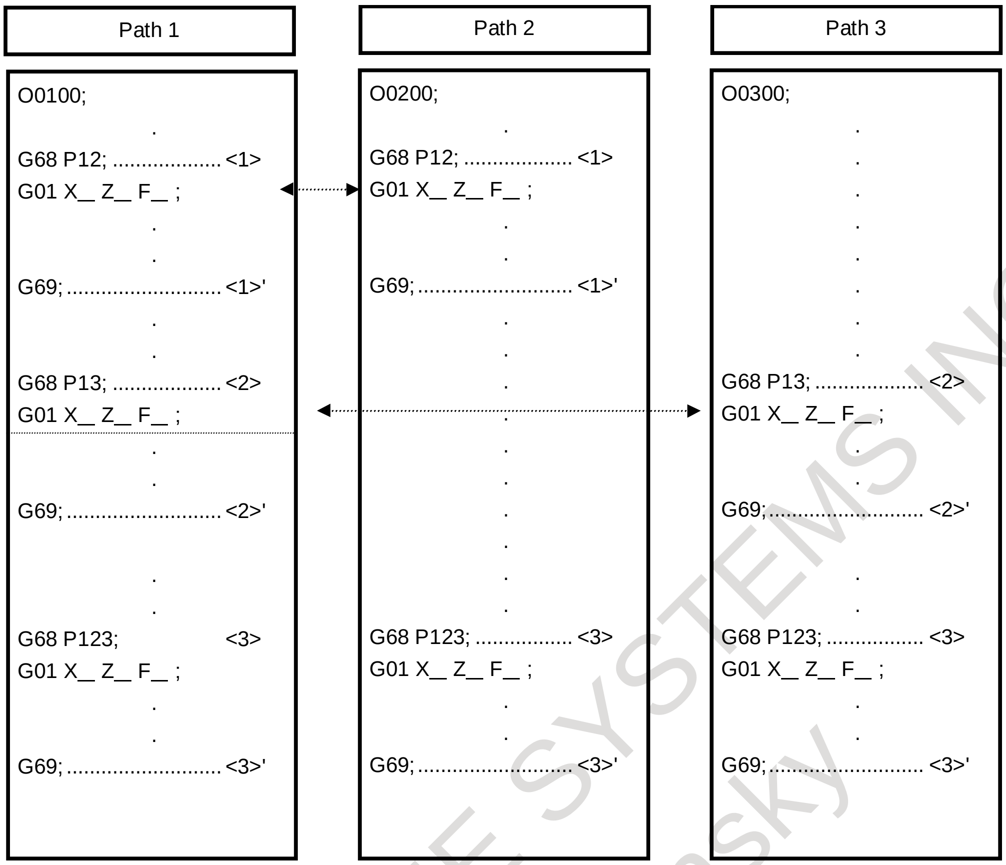
<1> G68 P3; (balance cut for paths 1 and 2)
Performs balance cutting for paths 1 and 2.
Balance cutting is performed according to the cutting feed commands between <1> and <1>'.

<2> G68 P5; (balance cut for paths 1 and 3)
Performs balance cutting for paths 1 and 3.
Balance cutting is performed according to the cutting feed commands between <2> and <2>'.

<3> G68 P7; (balance cut for paths 1, 2, and 3)
Performs balance cutting for paths 1, 2, and 3.
Balance cutting is performed according to the cutting feed commands between <3> and <3>'.

- When the value specified at P is obtained using path numbers in combination

Programs O100, O200, and O300 for individual paths are executed as follows:



<1> G68 P12; (balance cut for paths 1 and 2)
Performs balance cutting for paths 1 and 2.
Balance cutting is performed according to the cutting feed commands between <1> and <1>'.

<2> G68 P13; (balance cut for paths 1 and 3)
Performs balance cutting for paths 1 and 3.
Balance cutting is performed according to the cutting feed commands between <2> and <2>'.

<3> G68 P123; (balance cut for paths 1, 2, and 3)
Performs balance cutting for paths 1, 2, and 3.
Balance cutting is performed according to the cutting feed commands between <3> and <3>'.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8103							MWP	

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 MWP To specify a P command for the waiting M code/balance cut:
0: A binary value is used as conventionally done.
1: A path number combination is used.

Caution

> ⚠ **CAUTION**
> 1 Balance cuting only starts cutting feed on both tool posts at the same time; it does not maintain synchronization thereafter. To synchronize all the movements of both tool posts, the setting for both tool posts, such as the travel distance and feedrate, must be the same. Override and interlock can be applied independently to both tool posts. The settings for both tool posts that are related to override and interlock must also be the same to perform balance cutting.
> 2 After feed hold is applied during execution of balance cutting for both tool posts, balance cutting is not performed at the restart. Balance cutting is performed when the next move command is executed for both tool posts.

Note

> **NOTE**
> 1 Time delay before the pulse distribution of both tool posts is started is 2 msec or shorter.
> 2 Overlap is invalid. In the balance cut mode, synchronization is established at the start of each move block in which cutting is specified, so movement can momentarily stop.
> 3 In the balance cut mode, continuous thread cutting overlap is also invalid. Perform continuous thread cutting in the balance cut cancel mode.
> 4 To establish synchronization at the start of pulse distribution in a block in which thread cutting is specified, the same PC must be selected.
> 5 The cancel mode (G69) is unconditionally set by a reset.
> 6 When the mirror image for double turrets is selected, the balance cut function cannot be used.

Alarm and message

Number	Message	Description
PS0163	ILLEGAL COMMAND IN G68/G69	G68 and G69 are not independently commanded in balance cut. An illegal value is commanded in a balance cut combination (address P).

8.6 SYNCHRONOUS/COMPOSITE CONTROL

Overview

Multi-path control, which has multiple independent control paths built in, is used for such purposes as controlling multiple turrets of a complex lathe. The axes (such as X1-and Z1-axes) belonging to path 1 are controlled by commands in path 1, and the axes (such as X2- and Z2-axes) belonging to path 2 are controlled by commands in path 2.

- Independent control in each path

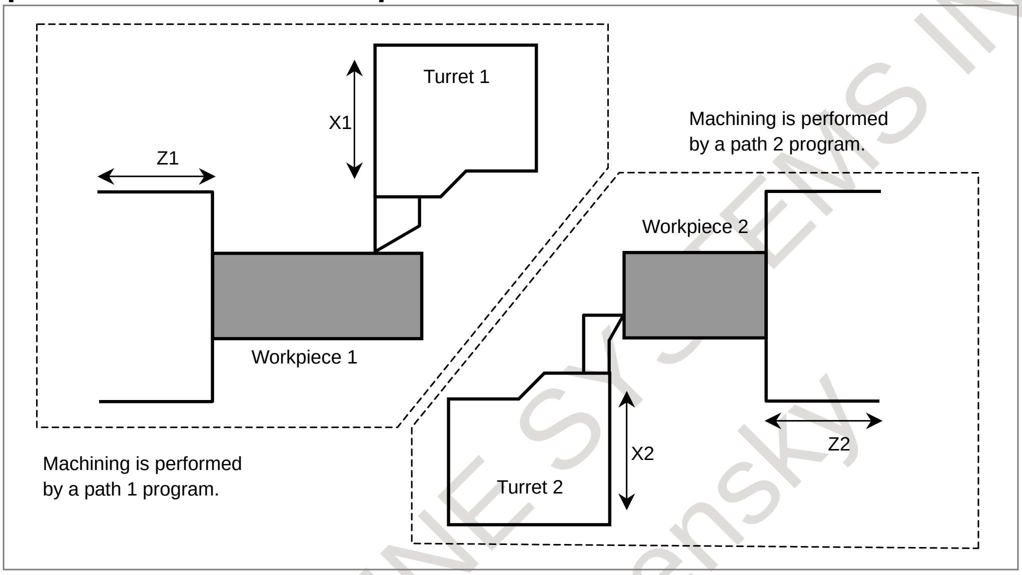


Fig. 8.6 (a)

This function enables synchronous control between paths or within a path, composite control between paths, and superimposed control between paths, as explained below.

- Synchronous control

- Synchronization of an axis in one path with an axis in another path
  (Example)
  Synchronization of the Z1-axis (master) with the Z2-axis (slave)

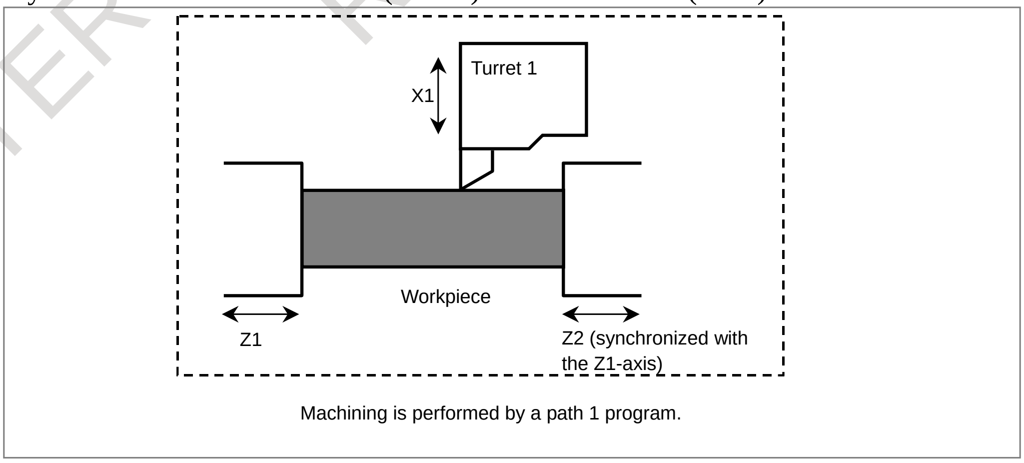


Fig. 8.6 (b)

- Synchronization of an axis in one path with another axis in the same path
  (Example)
  Synchronization of the Z1-axis (master) with the B1-axis (slave)

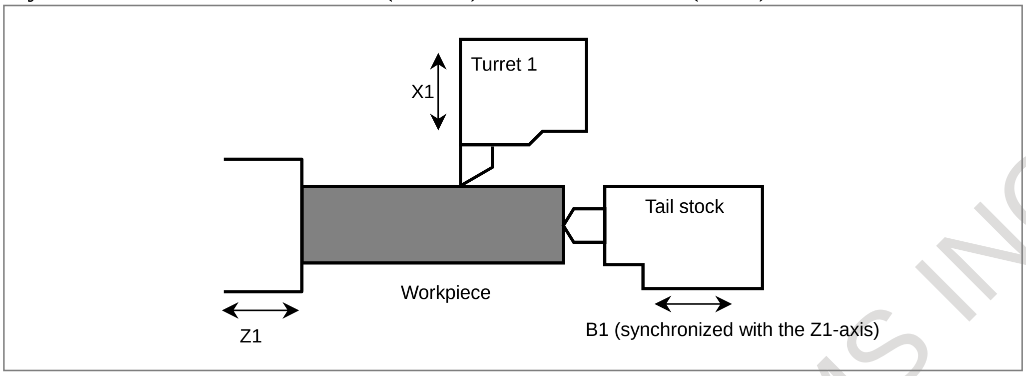


Fig. 8.6 (c)

- Composite control

- Interchanging move commands for an axis in one path with those for an axis in another path.
  (Example)
  Interchanging commands between the X1- and X2-axes
  → Control both X2- and Z1-axes by commands in a path 1 program
  Control both X1- and Z2-axes by commands in a path 2 program

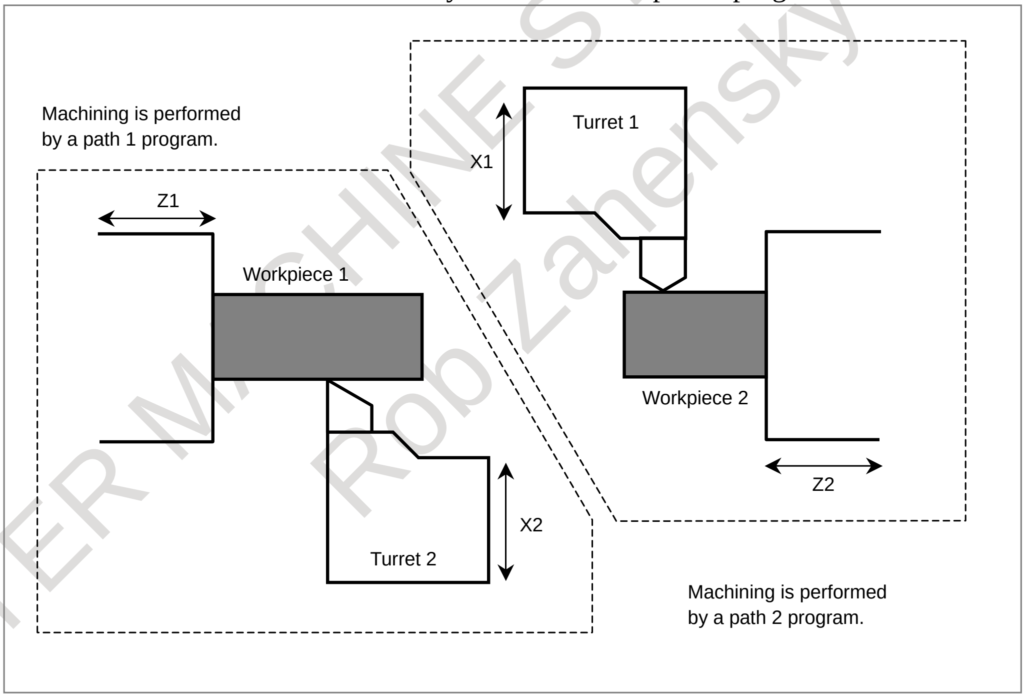


Fig. 8.6 (d)

8.6.1 Synchronous Control

An axis in one path can be synchronized with another axis in the same path or an axis in another path. This is done by issuing the same move commands for one axis (synchronous master axis) to another axis (synchronous slave axis). Using bit 0 (SMRx) of parameter No. 8162 can cause the slave axis to move in the direction opposite to that of the master axis. It is possible to place either the master or slave axis in a parking state, which means to discontinue giving move commands to a servo system. No coordinates are updated in the parking state. However, the absolute and relative coordinates can be updated using bit 2 (PKUx) of parameter No. 8162.

Explanation

- Setting

Specify which axis is to be the synchronous master axis, with parameter No. 8180, using the path number and the axis number.

(Example)

For an axis configuration in which all paths consist of X, Z, and Y axes

To synchronize the Z4-axis (slave) with the Z1-axis (master):

Parameter No. 8180z of path 4 = 102

To synchronize the X8-axis (slave) with the X5-axis (master):

Parameter No. 8180x of path 8 = 501

To synchronize the Y1-axis (slave) with the X1-axis (master):

Parameter No. 8180y of path 1 = 101

- Programming

Both before and after the M codes for a start and a cancellation of synchronous control, specify a waiting M code.

Master axis path	Slave axis path	
:	:	
M100P12;	M100P12;	Waiting
	M200;...........................	Start of synchronous control
M101P12;	M101P12;	Waiting
:	:	
Independent operation	Independent operation	
:	:	
M100P12;	M100P12;	Waiting
	M201;	Cancellation of synchronous control
M101P12;	M101P12;	Waiting
:	:	

- Signal operation

When synchronization begins or ends (when an M code is issued, for example), the synchronous control axis selection signals SYNC1 to SYNC8<Gn138> for the slave axis (from the PMC to the CNC) are changed from "0" to "1" (to begin synchronization) or from "1" to "0" (to terminate synchronization). To place an axis in a parking state, a parking signal PK1 to PK8<Gn122> is set to "1" for the target axis.

- Examples of applications

The following operations can be performed by using the synchronization functions together with the parking function, which causes move commands for an axis to be ignored and keeps the axis at a rest.

(1) Moving an axis in one path in synchronization with an axis in another path (Both master and slave axes move.)

Example 1)

Synchronizing the Z2-axis (slave) with the Z1-axis (master) (machining with both ends of a workpiece chucked)

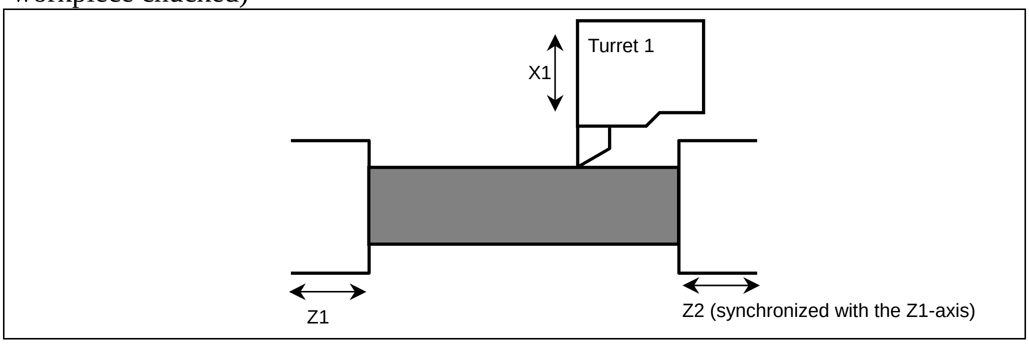


Fig. 8.6.1 (a)

Example 2)

Synchronizing the X2- and Z2-axes (master) with the X1- and Z1-axes (slave) (balance cutting)

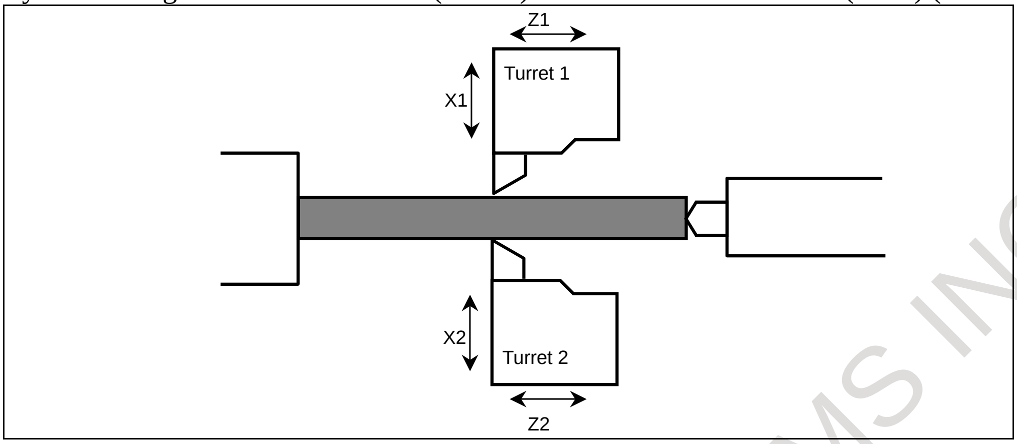


Fig. 8.6.1 (b)

Example 3)

Synchronizing the B1-axis (slave) (tail stock axis) with the Z1-axis (master)

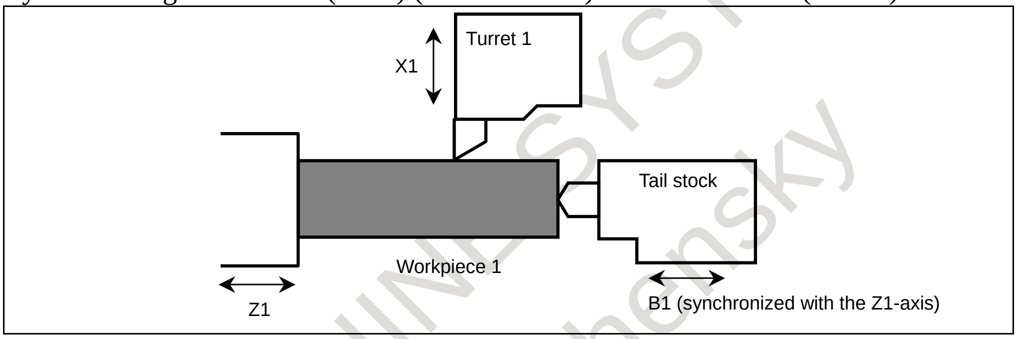


Fig. 8.6.1 (c)

(2) Placing the movement along the synchronous master axis in the stopped state using a parking signal is referred to as master parking. In this state, the tool will move along the slave axes only. In contrast, placing the synchronous slave axes in the stopped state is referred to as slave parking. In this state, the tool will move along the master axis only. This makes it possible to control a single motor from both paths 1 and 2.

Example 4)

Sharing one motor with the Z1- and Z2-axes (assuming that the motor is linked to the Z1-axis)

Master axis parking

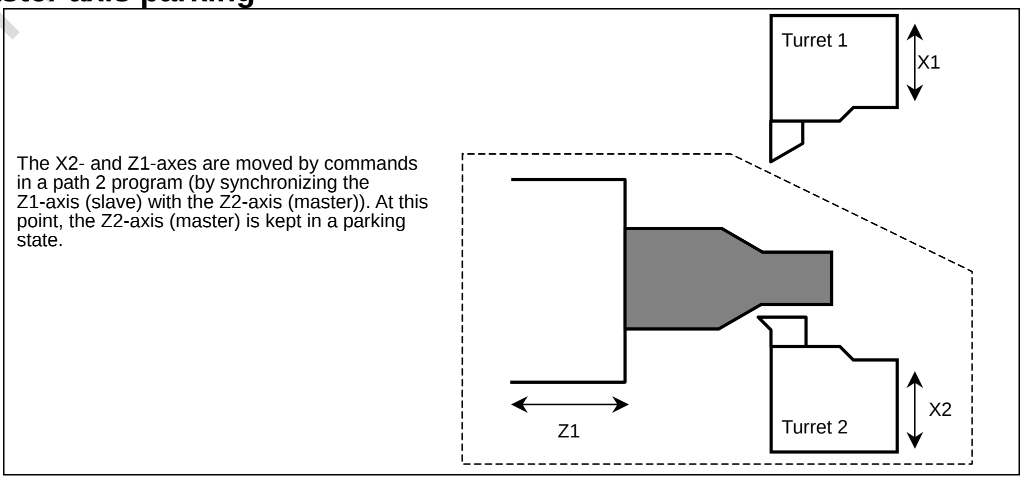


Fig. 8.6.1 (d)

Slave axis parking

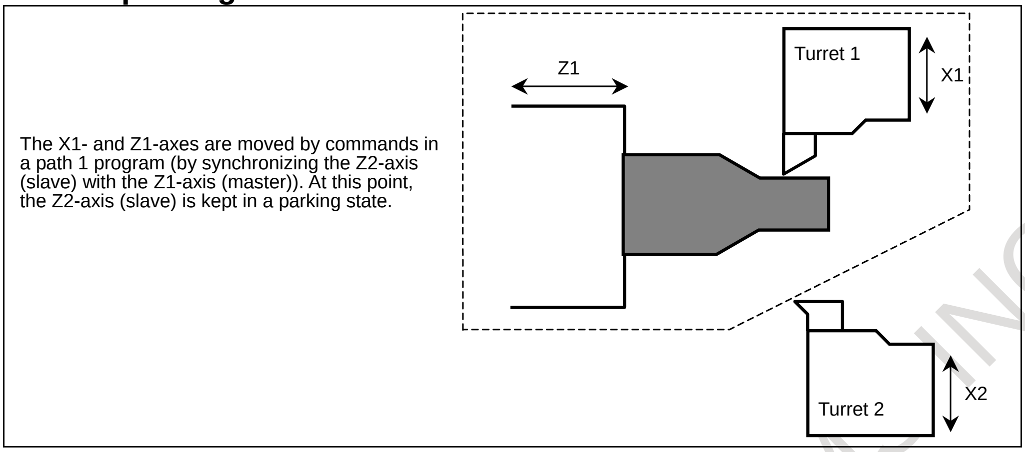


Fig. 8.6.1 (e)

Parking causes the positional relations between machine, absolute, and relative coordinates to shift. After reference position return, therefore, set the workpiece coordinate system.

- Reference position return and its check during synchronous control

If a reference position return command is issued for a synchronous master axis during synchronous control, it is executed normally for the master axis, but the slave axis does not return to its reference position (the slave axis only moves in synchronization with the reference position return of the master axis).
An exception is for automatic reference position return (G28) issued when the master axis is in a parking state, in which the amount of movement is calculated so that the slave axis returns to its reference position. In this case, a reference position return has been carried for the slave axis. If the reference position of the slave axis has not been established, alarm PS0354 “THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN SYNC MODE” will result.

If, in Cs axis synchronous control, the master axis is not in the parking state and the reference position of the master axis has not been established, a reference position return operation will be performed in reference to the master axis machine position, with the G28 command for the master axis.
If the master axis is in the parking state and the reference position of the slave axis has not been established, a reference position return operation will be performed in reference to the slave axis machine position, with the G28 command for the master axis.

If more than one slave axis belongs to one master axis, a reference position return command is executed so that the lowest-numbered slave axis returns to its reference position. If the master axis in one path is subjected to both synchronization with an axis in the same path and synchronization with an axis in the other path simultaneously, the lowest-numbered slave axis in the two paths is moved to the reference position.

A return to the second (third or fourth) reference position by the G30 command works in the same way as G28. In other words, usually only the master axis moves to the second (third or fourth) reference position. If the master axis is parking, the lowest-numbered axis is caused to move to its second (third or fourth) reference position.
If a reference position return check (G27) is specified during synchronous control, the master and slave axes move to the specified position. Upon completion of movement, a check is made to see whether the master axis is at its reference position (no check is made for the salve axes) unless the master axis is in a parking state, in which case a check is made upon completion of positioning to see whether the lowest-numbered slave axis is at its reference position.

- Out-of-synchronization detection

The term synchronous control used here only refers to an operation in which the same move command is issued to multiple different servo systems at one time. Note that synchronous control does not involve out-of-synchronization compensation, in which the positional deviation between multiple servo motors is constantly checked and one of the servo motors is subjected to compensation to reduce the deviation. However, using bit 1 (SERx) of parameter No. 8162 can specify detection of out-of-synchronization.
When the out-of-synchronization state is detected, the synchronization mode is canceled immediately using bit 6 (SESx) of parameter No. 8169, alarm SV0407, “EXCESS ERROR” is issued and the servo ready signal is turned off, or the excess synchronization error signal SEO <Fn559> is output.

- Automatic setting of a workpiece coordinate system

When synchronous control is started in a workpiece coordinate system, it is possible to specify the workpiece coordinate system automatically. When synchronous control for a workpiece coordinate system is terminated, it is possible to return the workpiece coordinate system to ordinary machining (not synchronous control). The explanation of the workpiece coordinate system used during synchronous control follows. When synchronous control is used to move an axis differently from the way originally specified, for example, the master axis may be placed in a parking state, while the slave axis is allowed to move. In such a case, it will be convenient if a coordinate system that indicates the current position of the slave axis is used as a workpiece coordinate system for the master axis. Conventionally, this workpiece coordinate system must be specified by program when synchronous control is started, because the workpiece coordinate system does not originally belong to the master axis. This automatic workpiece coordinate system setting function for synchronous control sets up this workpiece coordinate system automatically. This function can also resume the original workpiece coordinate system for the master axis automatically. Note that the workpiece coordinates of a salve axis can be automatically set through appropriate parameter setting at the end of synchronous control only.

- Setting and commands

In addition to setting ordinary synchronous control, parameters must be specified as follows:

(1) To set up a workpiece coordinate system for synchronous control automatically when starting synchronous control
Set bit 1 (SPMx) of parameter No. 8163 to 1.
Set parameter No. 8185 with the coordinates of the slave axis reference position relative to the coordinates of the master axis when the master axis is at the reference position.

(2) To resume the ordinary workpiece coordinate system automatically when terminating synchronous control
Set bit 2 (SPSx) of parameter No. 8163 to 1. (Only the parameter of the master axis.)
Set parameter No. 1250 with the master axis coordinates in the workpiece coordinate system when the master axis is at the reference position.
This synchronous control can be specified using the synchronous control axis selection signals (SYNC1 to SYNC8) similarly to the ordinary synchronous control. When the signals are raised to start synchronous control, a workpiece coordinate system for the master axis is automatically set up. When the signals are dropped to terminate synchronous control, the original workpiece coordinate system for the master axis is resumed automatically.

- Workpiece coordinate calculation method

(1) Workpiece coordinate system for synchronous control
(Master axis workpiece coordinate value) = (parameter No. 8185 for the master axis)
± (slave axis machine coordinate value) .... <1>
+ (master axis machine coordinate value)............ <2>

<1> +: Bit 4 (SCDx) of parameter No. 8163 for the master axis = 0
-: Bit 4 (SCDx) of parameter No. 8163 for the master axis = 1

<2> Bit 3 (SCMx) of parameter No. 8163 for the master axis = 1 only

(2) Workpiece coordinate system for ordinary operation
(Master axis workpiece coordinate value)
= (parameter No. 1250 for the master axis) +(master axis machine coordinate value)

Caution

⚠ CAUTION
The same least command and input increments must apply to both master and slave axes.

Note

NOTE

1. If more than one slave axis is synchronized with one master axis, the master axis is set with the workpiece coordinate system that corresponds to the current position of the first slave axis that is synchronized with the master axis.
2. A coordinate system can also be set with consideration given to tool offset, through appropriate parameter settings. So, the coordinate system is set up normally even when tool geometry compensation is applied.

8.6.2 Composite Control

Move commands can be interchanged between an axis in one path and an axis in another path. In other words, when a machining program is executed for one path, actual machining can be performed with an axis in the other path. Coordinate systems can also be switched automatically between independent control and composite control.

Explanation

- Setting

Specify which axis is to interchange with which axis, with parameter No. 8183, using the path numbers and the axis numbers.

Example)

For an axis configuration in which all paths consist of X, Z, and Y axes
Between the Z1- and Z4-axes:

Parameter No. 8183z of path 4 = 102	Between the X5- and X8-axes:
Parameter No. 8183x of path 8 = 501	Between the X1- and Y2-axes:
Parameter No. 8183y of path 2 = 101	

To set up coordinate systems automatically when composite control begins or ends, set bits 4 (MPMx) and 5 (MPSx) of parameter No. 8162 to 1, and specify the positional relationship between the coordinate systems in parameter No. 8184.

- Programming

Both before and after the M codes for a start and a cancellation of composite control, specify a waiting M code.

Composite control source	Composite control destination
:	:
M100P12;	M100P12; ....Waiting
	M210;....................................Start of composite control
M101P12;	M101P12; ....Waiting
:	:
Independent operation	Independent operation
:	:

```
M100P12;            M100P12; ...............................Waiting
                    M211;  ..................................Cancellation of composite control
M101P12;            M101P12; ...............................Waiting
  :                   :
```

- Signal operation

When you want to start or cancel composite control (when you specify an M code, for example), change the composite control axis select signal between MIX1 and MIX8<Gn128> for the axis subject to composite control that is specified with parameter No. 8183 (the signal being sent from the PMC to the CNC) from "0" to "1" (to start composite control) and from "1" to "0" (to cancel composite control).

- Examples of applications

Suppose that a machine has the X1- and Z1-axes belonging to path 1 and the X2- and Z2-axes belonging to path 2 and that a workpiece moves along the Z1- and Z2-axes as directed by move commands. The following examples interchange commands between the X1- and X2-axes.

(1) Independent control

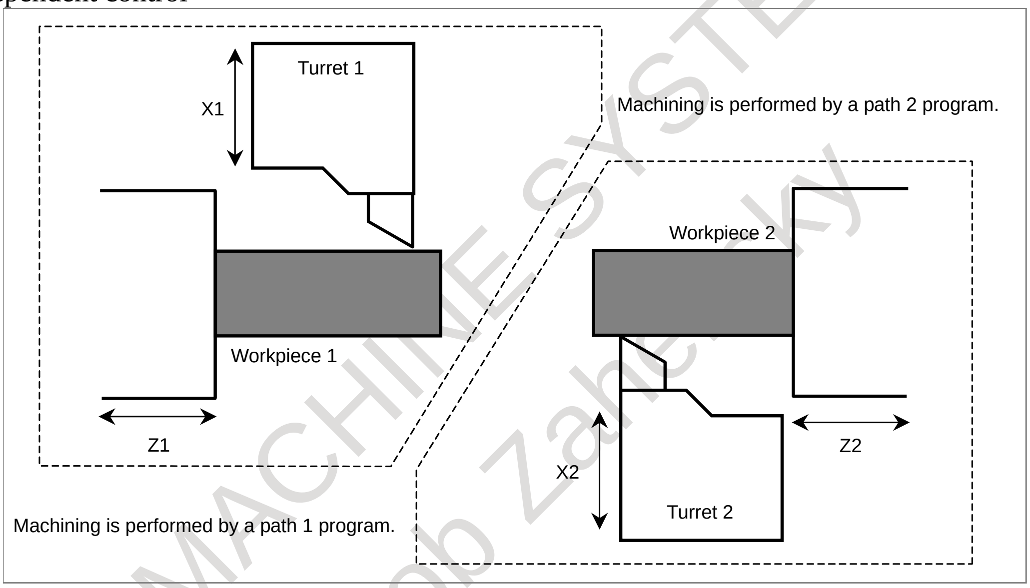


Fig. 8.6.2 (a)

(2) Composite control

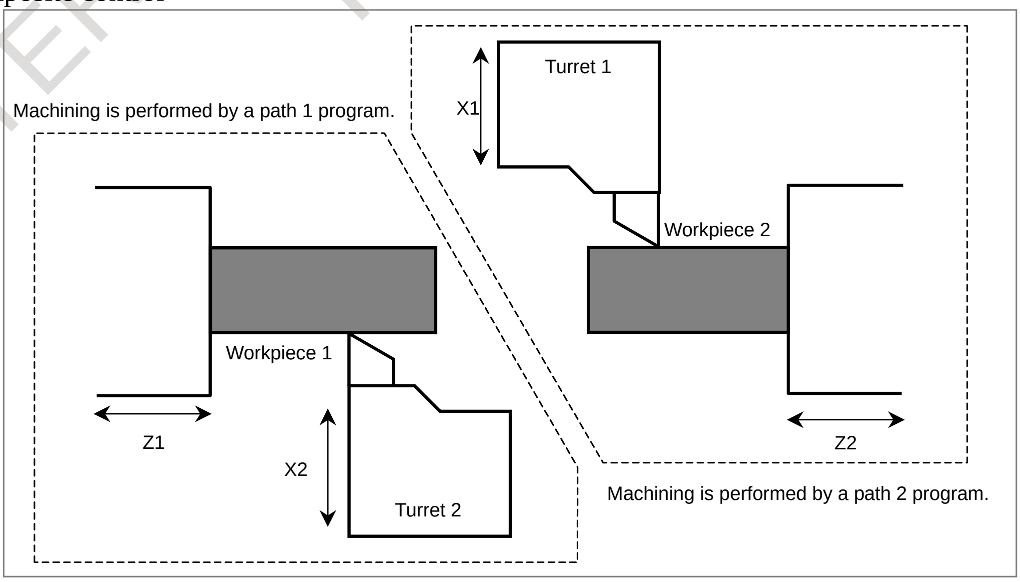


Fig. 8.6.2 (b)

During composite control, the X2- and Z1-axes are moved by a path 1 program, and the workpiece coordinates of the X-axis in path 1 indicates the position of turret 2. Similarly, the X1- and Z2-axes are moved by a path 2 program, and the workpiece coordinates of the X-axis in path 2 indicates the position of turret 1.

- Spindle control

The composite control function does not switch the spindle speed command or the feed per rotation command based on feedback pulses from the position coder. Therefore, the spindle speed command and feedback pulses should be switched using the spindle command selection signals and spindle feedback selection signals.

- Tool offset during composite control

At the time of a switch between independent control and composite control, the preset offset value, tool nose radius compensation/cutter compensation/tool length amounts will not be changed. After a control switch, you must issue a new T/D/H code command to set an offset value again, if required.

- Reference position return during composite control

If G28 is issued to specify an automatic reference position return for an axis in one path during composite control, an amount of movement is calculated so that the associated axis in the other path can move to the reference position. In this case, the reference position for that axis must have already been established. If the reference position of the axis of the other path subject to composite control has not been established, alarm PS0359 “THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN COMP MODE” will result.
A manual reference position return is not allowed.
However, that during Cs axis composite control, a manual reference position return is allowed. If, after the establishment of a reference position and before a switch to composite control, you want to enter the state in which no reference position is established, set bit 5 (CRZ) of parameter No. 8161 to 1.

- Automatic workpiece coordinate system setting

By parameter setting, a workpiece coordinate system to be used during composite control can be automatically set when composite control is started. When composite control is terminated, the workpiece coordinate system can be automatically returned to the workpiece coordinate system used for ordinary machining not under composite control.
A workpiece coordinate system to be used during composite control is explained below. For example, when a movement is made on axes other than the specified axes by using composite control, it is convenient to use a coordinate system that indicates the current position on the movement axes, as a workpiece coordinate system for the specified axes. Such a workpiece coordinate system differs from the original workpiece coordinate system for the specified axes, so that the coordinate system needs to be set by programming when composite control is started. Moreover, the workpiece coordinate system needs to be returned to the original workpiece coordinate system for the specified axes when composite control is terminated. With this function, these coordinate systems can be set automatically.

Setting, command

When composite control is started, a workpiece coordinate system to be used during composite control is automatically set for axes with bit 4 (MPMx) of parameter No. 8162 set to 1. When composite control is terminated, the workpiece coordinate system can be automatically returned to the workpiece coordinate system used for ordinary machining not under composite control in connection with the axes with bit 5 (MPSx) of parameter No. 8162 set to 1.

To specify this function, the composite control axis change selection signals MIX1 to MIX32 are used as in the case of ordinary composite control. When composite control is started by turning on signals, a workpiece coordinate system for the composite axes is automatically set. Similarly, when composite control is canceled by turning off signals, a workpiece coordinate system for the composite axes is automatically set.

Workpiece coordinate calculation method

1) When no workpiece coordinate system (G54 to G59, including additional workpiece coordinate systems) is used, a coordinate value calculated from the coordinate value (parameter No. 8184) of the reference position on the composite control target axis in the workpiece coordinate system of the local axis and the machine coordinate value of the composite control target axis is set in a coordinate system at composite control start time.
In a coordinate system at composite control termination time, a coordinate value calculated from the coordinate value (parameter No. 1250) of the reference position used for automatic coordinate system setting on the local axis and the machine coordinate value on the local axis is set.
The method of coordinate value calculation is described below.

(Example)

Composite control where the X1-axis and X2-axis are replaced with each other

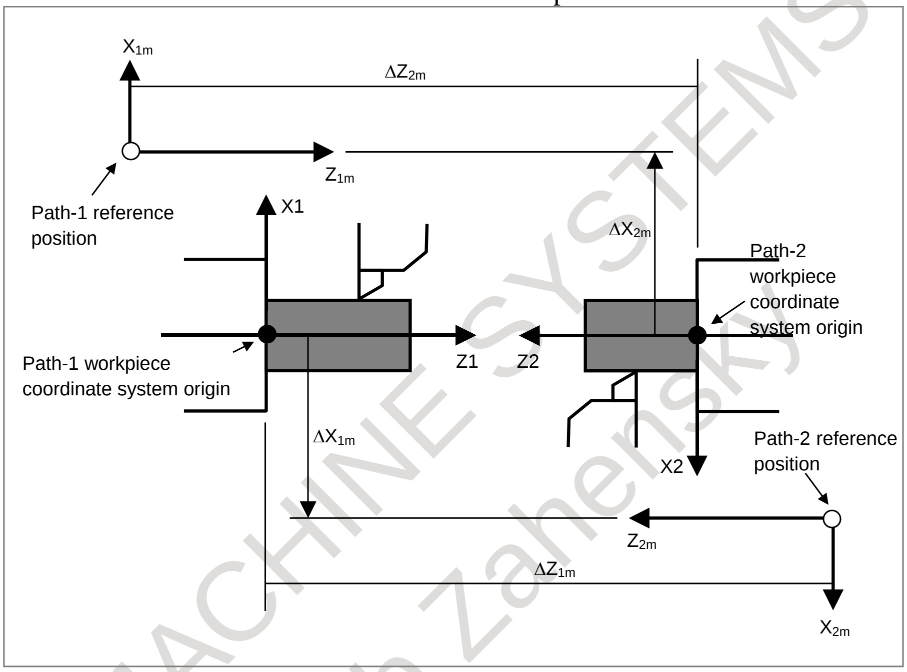


Fig. 8.6.2 (c)

The reference position of path 2 in the workpiece coordinate system of path 1 is at (ΔX1m,ΔZ1m). The reference position of path 1 in the workpiece coordinate system of path 2 is at (ΔX2m,ΔZ2m). Set ΔX1m in parameter No. 8184x for path 1. Set ΔX2m in parameter No. 8184x for path 2.

When composite control is started, a workpiece coordinate system is set to satisfy the following:
X1 = (Path-1 X-axis setting) ± (X2 machine coordinate value)
+ when bit 6 (MCDx) of parameter No. 8162 for path 1 = 0
– when bit 6 (MCDx) of parameter No. 8162 for path 1 = 1
X2 = (Path 2 X-axis setting) ± (X1 machine coordinate value)
+ when bit 6 (MCDx) of parameter No. 8162 for path 2 = 0
– when bit 6 (MCDx) of parameter No. 8162 for path 2 = 1

When composite control is terminated, a workpiece coordinate system is set to satisfy the following:
X1 = (Parameter No. 1250 for path 1) + (X1 machine coordinate value)
X2 = (Parameter No. 1250 for path 2) + (X2 machine coordinate value)

2) When a workpiece coordinate system (G54 to G59, including additional workpiece coordinate systems) is used, workpiece coordinate system preset operation (equivalent to G92.1 IP 0) is used for setting as described below, instead of coordinate values based on the calculation method of 1) above.
   When composite control is started, the workpiece coordinate system based on specified axes is preset to a workpiece coordinate system shifted on the specified axes by the currently valid workpiece origin offset values from the machine origin on the movement axes.
   When composite control is terminated, the workpiece coordinate system based on composite axes is preset to a workpiece coordinate system shifted on the local axes by the currently valid workpiece origin offset values from the machine origin on the local axes.

- Composite control between Cs contour control axes

When bit 4 (SMT) of parameter No. 8165 is set to 1, when composite control is executed between axes of Cs contour control, torque limit skipping is possible in the state of torque limit command signal TLMH and load detection signal LDT1 of the Cs contour control axis of the composite control destination.
The torque limit skip command (G31 P98 / P99) is commanded by composite control source.
For details of the torque limit skip for the Cs contour control, refer to "Cs Contour Control Torque Limit Skip".

For signal TLMH and signal LDT1, refer torelevant manuals such as "FANUC Spindle Motor α *i* series PARAMETER MANUAL" (B-65280EN).

Signals

Synchronous control axis selection signals SYNC1 to SYNC8<Gn138>

[Classification] Input signal
[Function] These signals perform synchronous control.
[Operation] When one of these signals becomes "1", the control unit:
Begins synchronous control in such a way that the corresponding axis becomes a slave axis.
The axis with which the slave axis is synchronized is determined by parameter No. 8180.

Parking signals PK1 to PK8<Gn122>

[Classification] Input signal
[Function] These signals place each axis in a parking state.
[Operation] When one of these signals becomes "1", the control unit:
- Places the corresponding axis in a parking state.
  If the corresponding axis is under synchronous control, it enters a parking state immediately regardless of whether the axis is moving. If a parking signal is set to "1" without specifying synchronous control, it is ignored.

Composite control axis change selection signals MIX1 to MIX8<Gn128>

[Classification] Input signal
[Function] These signals perform composite control.
[Operation] When one of these signals becomes "1", the control unit:
- Begins composite control over the corresponding axis.
  The axis with which the corresponding axis is controlled together is determined by parameter No. 8183.

> **NOTE**
> For bit 1 (MIX) of parameter No. 8166, only a single signal for path 1 is available.

Synchronous/composite/superimposed control under way signals SYN1O to SYN8O<Fn118>

[Classification] Output signal

[Function] These signals indicate each axis is being subjected to ssynchronous, composite, or superimposed control.

[Output cond.] These signals become "1" under the following condition:

- The corresponding axis is under synchronous, composite, or superimposed control.

These signals become "0" under the following condition:

- The corresponding axis is not under synchronous, composite, or superimposed control.

> ⚠ **CAUTION**
> Whether each axis is under synchronous or composite control does not always match whether the corresponding selection signal (synchronous control axis selection or composite control axis selection) has been issued or not. For example, if these signals are set to "1" during an alarm, they are ignored. If a servo alarm occurs during these types of control, they are terminated automatically. Before attempting to perform these types of control, always check the state of these signals.

Synchronous master axis confirmation signals SYCM1 to SYCM8<Fn341>

[Classification] Output signal

[Function] These signals notify whether the corresponding axes are synchronous master axes.

[Output cond.] These signals become "1" under the following condition:

- The corresponding axes are synchronous master axes.

These signals become "0" under the following condition:

- The corresponding axes are released from synchronous control.

Synchronous slave axis confirmation signals SYCS1 to SYCS8<Fn342>

[Classification] Output signal

[Function] These signals notify whether the corresponding axes are synchronous slave axes.

[Output cond.] These signals become "1" under the following condition:

- The corresponding axes are synchronous slave axes.

These signals become "0" under the following condition:

- The corresponding axes are released from synchronous control.

Composite axis confirmation signals MIXO1 to MIXO8<Fn343>

[Classification] Output signal

[Function] These signals notify whether the corresponding axes are composite control axes.

[Output cond.] These signals become "1" under the following condition:

- The corresponding axes are composite control axes.

These signals become "0" under the following condition:

- The corresponding axes are released from composite control.

Parking axis confirmation signals SMPK1 to SMPK8<Fn346>

[Classification] Output signal

[Function] These signals notify whether the corresponding axes are parking axes in synchronous control.

[Output cond.] These signals become "1" under the following condition:

- The corresponding axes are parking axes in synchronous control.

These signals become "0" under the following condition:

- The corresponding axes are released from synchronous control or from parking.

Excess synchronization error signals SEO1 to SEO8<Fn559>

[Classification] Output signal
[Function] These signals indicate whether excess synchronization error is detected.
[Operation] These signals become "1" under the following condition:
- Synchronization error exceeds the allowable range specified in parameter No. 8181.

These signals become "0" under the following condition:
- Synchronization error is within the allowable range specified in parameter No. 8181.

NOTE
1 These signals are valid when bit 1 (SERx) of parameter No. 8162 is set to 1 and bit 6 (SESx) of parameter No. 8169 is set to 1.
2 These signals are output to the slave axis.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn122	PK8	PK7	PK6	PK5	PK4	PK3	PK2	PK1
Gn128	MIX8	MIX7	MIX6	MIX5	MIX4	MIX3	MIX2	MIX1
Gn138	SYNC8	SYNC7	SYNC6	SYNC5	SYNC4	SYNC3	SYNC2	SYNC1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn118	SYN8O	SYN7O	SYN6O	SYN5O	SYN4O	SYN3O	SYN2O	SYN1O
Fn341	SYCM8	SYCM7	SYCM6	SYCM5	SYCM4	SYCM3	SYCM2	SYCM1
Fn342	SYCS8	SYCS7	SYCS6	SYCS5	SYCS4	SYCS3	SYCS2	SYCS1
Fn343	MIXO8	MIXO7	MIXO6	MIXO5	MIXO4	MIXO3	MIXO2	MIXO1
Fn346	SMPK8	SMPK7	SMPK6	SMPK5	SMPK4	SMPK3	SMPK2	SMPK1
Fn559	SEO8	SEO7	SEO6	SEO5	SEO4	SEO3	SEO2	SEO1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8160	NRS	SPE	NCS					

[Input type] Parameter input
[Data type] Bit path

#5 NCS If an overtravel alarm is issued for an axis under synchronous, composite, or superimposed control, synchronous, composite, or superimposed control is:
0: Released.
1: Not released.

NOTE
If this parameter is 1 for any one of the paths in a machine group, it is assumed to be 1 for all the paths.

#6 SPE The synchronization deviation is:
0: The difference between the positioning deviation of the master axis and that of the slave axis.

1: The difference between the positioning deviation of the master axis and that of the slave axis plus the acceleration/deceleration delay.

> **NOTE**
> 1 When the master and slave axes have different acceleration/deceleration time constants, set 1.
> 2 SPE is valid when bit 1 (SERx) of parameter No. 8162 is set to 1. SPE is used to find a synchronization deviation for comparison with parameter No. 8181.

#7 NRS When the system is reset, synchronous, composite, or superimposed control is:
0: Released.
1: Not released.

	#7	#6	#5	#4	#3	#2	#1	#0
8161	NSR		CRZ					NMR

[Input type] Parameter input
[Data type] Bit

#0 NMR When an axis subject to composite control is placed in servo-off state:
0: Composite control is stopped
1: Composite control is not stopped, provided bit 0 (FUP) of parameter No. 1819 is set to 1 to disable follow-up for the axis.

#5 CRZ If the state of the composite control signal is switched in composite control on two axes under Cs contour control, the reference position establishment state of the two axes in composite control is:
0: Maintained. (The unestablished state is not assumed.)
1: Assumed to be unestablished.

#7 NSR When servo-off occurs with an axis in synchronous control:
0: Synchronous control is canceled.
1: Synchronous control is not canceled if follow-up operation is disabled for the axis (with bit 0 (FUPx) of parameter No. 1819 set to 1).

	#7	#6	#5	#4	#3	#2	#1	#0
8162	MUMx	MCDx	MPSx	MPMx		PKUx	SERx	SMRx

[Input type] Parameter input
[Data type] Bit axis

#0 SMRx Synchronous mirror-image control is:
0: Not applied. (The master and slave axes move in the same direction.)
1: Applied. (The master and slave axes move in opposite directions.)

#1 SERx The synchronization deviation is:
0: Not detected.
1: Detected.

> **NOTE**
> When both master and slave axes move in synchronization, the positioning deviations of the corresponding axes are compared with each other. If the difference is greater than or equal to the value specified in parameter No. 8181, an alarm occurs. When either axis is in the parking or machine-locked state, however, the synchronization deviation is not detected.

#2 PKUx In the parking state,

0: The absolute, relative, and machine coordinates are not updated.
1: The absolute and relative coordinates are updated. The machine coordinates are not updated.

> **NOTE**
> 1 With an axis for which polar coordinate interpolation (T series) is specified, set this parameter to 1. If this parameter is set to 0, a coordinate shift can occur when a single block stop or feed hold is performed in the polar coordinate interpolation mode (T series).
> 2 With an axis that is set to function as a synchronous master axis and synchronous slave axis at the same time (with bit 1 (SYWx) of parameter No. 8167), set this parameter to 1.
> 3 With an axis specified in the 3-dimensional coordinate conversion mode, set this parameter to 1. If this parameter is set to 0, the alarm PS0367 ”3-D CONV. WAS COMMANDED IN SYNC MODE AS THE PARAMETER PKUx(NO.8162#2) IS 0.” is issued.
> 4 When specify the G53 command during master parking, set this parameter to 1.

#4 MPMx When composite control is started, the workpiece coordinate system is:

0: Not set automatically.
1: Set automatically.

> **NOTE**
> When the workpiece coordinate system is automatically set at the start of composite control, it is calculated from the following:
> Current machine coordinates and the workpiece coordinates at the reference point of each axis (parameter No. 8184).
> When a workpiece coordinate system (G54 to G59, including additional workpiece coordinate systems) is used, however, instead of the coordinate value obtained by the above calculation, the workpiece coordinate value obtained by workpiece coordinate system presetting (equivalent to G92.1 IP 0) in the machine coordinate system of the other axis in composite control is set.

#5 MPSx When composite control is terminated, the workpiece coordinate system is:

0: Not set automatically.
1: Set automatically.

> **NOTE**
> When the workpiece coordinate system is automatically set at the end of composite control, it is calculated from the following: Current machine coordinates and the workpiece coordinates at the reference point of each axis under composite control (parameter No. 1250)
> When a workpiece coordinate system (G54 to G59, including additional workpiece coordinate systems) is used, however, instead of the coordinate value obtained by the above calculation, the workpiece coordinate value obtained by workpiece coordinate system presetting (equivalent to G92.1 IP 0) in the machine coordinate system of the local axis is set.

#6 MCDx The axes to be replaced with each other under composite control have the coordinate systems placed:

0: In the same direction. Simple composite control is applied. (A movement is made in the same direction along the corresponding axis.)

1: In opposite directions. Mirror-image composite control is applied. (A movement is made in the reverse direction along the corresponding axis.)

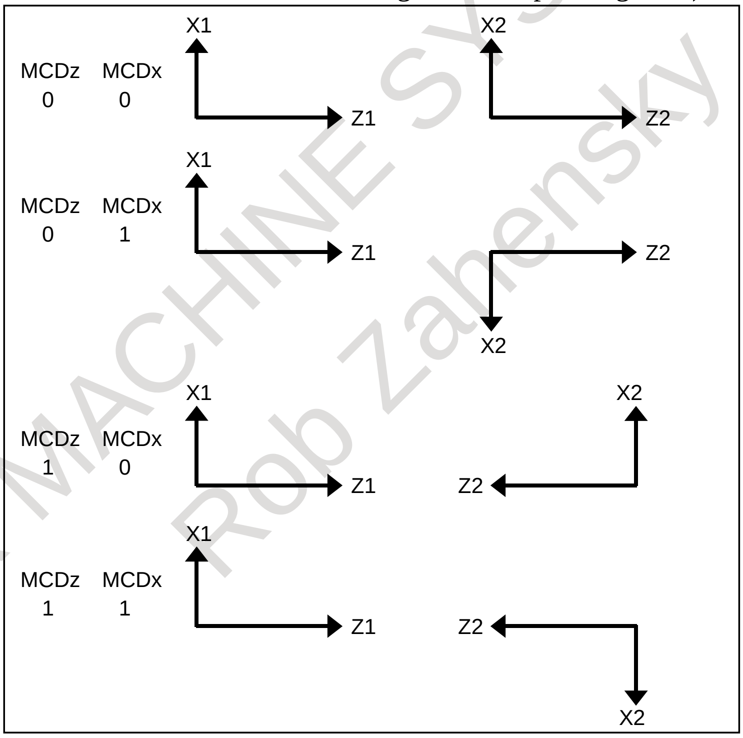


Fig. 8.6.2 (d)

#7 MUMx In composite control, a move command for the axis:

0: Can be specified.

1: Cannot be specified.

> **NOTE**
> Upon the execution of a move command along an axis for which MUMx is set to 1 during mixed control, alarm PS0353 is issued. For example, when axis X1 and axis X2 are placed under composite control, and a command for axis X2 (motor for axis X1) is to be disabled, set MUMx for path 2 to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
8163	NUMx	MMIx	SMIx	SCDx	SCMx	SPSx	SPMx	

[Input type] Parameter input
[Data type] Bit axis

#1 SPMx When synchronous control is started, automatic workpiece coordinate system setting for the master axis is
0: Not Performed.
1: Performed.

> **NOTE**
> When a workpiece coordinate system is automatically set at the start of synchronous control, the workpiece coordinate system is calculated from the current machine coordinates and the workpiece coordinates of each axis at the reference position set in parameter No. 8185.

#2 SPSx When synchronous control terminates, automatic workpiece coordinate system setting for the master axis is:
0: Not performed.
1: Performed.

> **NOTE**
> When a workpiece coordinate system is automatically set at the end of synchronous control, the workpiece coordinate system is calculated from the current machine coordinates and the workpiece coordinates for each axis at the reference position set in parameter No. 1250.

#3 SCMx When workpiece coordinates are calculated in synchronous control:
0: The workpiece coordinates are calculated from the machine coordinates of the slave axis.
1: The workpiece coordinates are calculated from the machine coordinates of the master axis and slave axis.

#4 SCDx The positive (+) directions of the master axis and slave axis in the coordinate system in synchronous control are:
0: Identical.
1: Opposite.
Set the parameters SPMx, SPSx, SCMx, and SCDx for the master axis. These settings are referenced during automatic workpiece coordinate setting for the master axis at the start of synchronous control.

#5 SMIx In synchronous control, the manual handle interruption amount for the master axis or the mirror image mode is:
0: Reflected in the slave axis.
1: Not reflected in the slave axis.

When this bit (SMIx) is set to 0
Manual handle interruption :
To the travel distance along the slave axis, the interruption amount of the master axis is also added.
Mirror image :
When mirror image is applied to the master axis, mirror image is also applied to the slave axis.

When this bit (SMIx) is set to 1
Manual handle interruption :
To the travel distance along the slave axis, the interruption amount of the master axis is not added.
Mirror image :
Even when mirror image is applied to the master axis, mirror image is not applied to the slave axis.

#6 MMIx For a composite control axis, manual handle interruption under composite control is:
0: Enabled.
1: Disabled.

#7 NUMx When neither synchronous control nor composite control is applied, a move command for the axis is:
0: Not disabled.
1: Disabled.

> **NOTE**
> If a move command is specified for an axis with NUMx set to 1 when neither synchronous control nor composite control is applied, alarm PS0353 is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
8164					MCEx	MCSx	MWEx	MWSx
					MCEx	MCSx		

[Input type] Parameter input
[Data type] Bit axis

#0 MWSx In automatic workpiece coordinate system setting, performed when composite control is started, a workpiece shift and position offset are:
0: Not considered.
1: Considered.

> **NOTE**
> When bit 4 (MPMx) of parameter No. 8162 is set to 1 and workpiece coordinate system (G54 to G59, including additional workpiece coordinate system) is not used, MWSx is enabled.

#1 MWEx In automatic workpiece coordinate system setting, performed when composite control is canceled, a workpiece shift and position offset are:
0: Not considered.
1: Considered.

> **NOTE**
> When bit 5 (MPSx) of parameter No. 8162 is set to 1 and workpiece coordinate system (G54 to G59, including additional workpiece coordinate system) is not used, MWEx is enabled.

#2 MCSx In automatic workpiece coordinate system setting, performed when composite control is started:
0: Parameter No. 8184 and the machine coordinate system of the composite control target path are used.
1: The absolute coordinate system of the composite control target path is used.

> **NOTE**
> When bit 4 (MPMx) of parameter No. 8162 is set to 1 and workpiece coordinate system (G54 to G59, including additional workpiece coordinate system) is not used, MCSx is enabled.

#3 MCEx In automatic workpiece coordinate system setting, performed when composite control is canceled:
0: Parameter No. 1250 and the machine coordinate system of the composite control target path are used.
1: The absolute coordinate system of the composite control target path is used.

> **NOTE**
> When bit 5 (MPSx) of parameter No. 8162 is set to 1 and workpiece coordinate system (G54 to G59, including additional workpiece coordinate system) is not used, MCEx is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
8165				SMT				

[Input type] Parameter input
[Data type] Bit

#4 SMT In composite control between Cs contour control axes, torque limit skip is:
0: Disabled
1: Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
8166							MIX	

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 MIX For composite control:
0: An interface for three paths or more is used. In this case, set the composite control axis selection signals MIX1 to MIX8 for the axis that is placed under composite control by parameter No. 8183, from “0” to “1” or from “1” to “0”.

1: The conventional 2-path interface is used. (Composite control on three paths or more is disabled.) In this case, set parameter No. 8183 for path 2, and use the composite control axis selection signals of path 1.

	#7	#6	#5	#4	#3	#2	#1	#0
8167		SPVx	SWSx	SWMx	SGSx	SGMx	SYWx	
		SPVx					SYWx	

[Input type] Parameter input
[Data type] Bit axis

#1 SYWx The axis is:
0: Not used as a master axis and slave axis at the same time.
1: Used as a master axis and slave axis at the same time.

#2 SGMx In automatic workpiece coordinate system setting at the start of synchronous control, a tool offset is:
0: Considered.
1: Not considered.

> **NOTE**
> SGMx is enabled when bit 1 (SPMx) of parameter No. 8163 is set to 1.

#3 SGSx In automatic workpiece coordinate system setting at the end of synchronous control, a tool offset is:
0: Considered.
1: Not considered.

> **NOTE**
> SGSx is enabled when bit 2 (SPSx) of parameter No. 8163 or bit 6 (SPVx) of parameter No. 8167 is set to 1.

#4 SWMx In automatic workpiece coordinate system setting at the start of synchronous control, a workpiece shift is:
0: Not considered.
1: Considered.

> **NOTE**
> SWMx is enabled when bit 1 (SPMx) of parameter No. 8163 is set to 1.

#5 SWSx In automatic workpiece coordinate system setting at the end of synchronous control, a workpiece shift is:
0: Not considered.
1: Considered.

> **NOTE**
> SWSx is enabled when bit 2 (SPSx) of parameter No. 8163 or bit 6 (SPVx) of parameter No. 8167 is set to 1.

#6 SPVx At the end of synchronous control, automatic workpiece coordinate system setting for the slave axis is:
0: Not performed.
1: Performed.

> **NOTE**
> When a workpiece coordinate system is automatically set at the end of synchronous control, the workpiece coordinate system is calculated from the current machine coordinates and the workpiece coordinates for each axis at the reference position set in parameter No. 1250.

	#7	#6	#5	#4	#3	#2	#1	#0
8168		WST		MWR		SVF	MSO	MPA

[Input type] Parameter input
[Data type] Bit

#0 MPA If an alarm concerning synchronous control, composite control, or superimposed control is issued:
0: All paths of the machine group to which the alarm occurrence path belongs are placed in feed hold state.
1: Only the path including the axis placed under synchronous control, composite control, or superimposed control is placed in the feed hold state.

#1 MSO When one of the following events occurs in synchronous control, composite control or superimposed:
- The emergency stop signal *ESP <Gn008.4> is turned off.
- The servo-off signals SVF1 to SVF8 <Gn126.0 to Gn126.7> are turned on.
- A servo alarm is issued.

0: The synchronous, composite control or superimposed control mode is canceled and follow-up operation is not performed.
For the operation to be performed when the servo-off signal is turned on, however, the setting of bit 7 (NSR) of parameter No. 8161 is used in synchronous control or the setting of bit 0 (NMR) of parameter No. 8161 is used in composite control.
1: The synchronous, composite control or superimposed control mode is not canceled.
The following operation is performed to perform follow-up operation:
When the emergency stop signal *ESP is turned off, the relevant path is determined and operation is performed so that the emergency stop signal *ESP is virtually turned off for the determined path.
When the servo-off signals SVF1 to SVF8 are turned on, the relevant axis is determined and operation is performed so that the servo-off signals SVF1 to SVF8 are virtually turned on for the determined axis.
When a servo alarm is issued, the relevant axis is determined and the alarm SV0003, “SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL MODE CAN'T BE CONTINUED” is issued for the determined axis to stop moving the tool along the axis. When bit 2 (SVF) of parameter No. 8168 is set to 1, this servo-off specification follows the SVF setting.

> **NOTE**
> 1 This setting is valid also during operation. For all axes placed under synchronous, composite control or superimposed control, the emergency stop signal is turned off, the servo-off signal is turned on, or a servo alarm is issued.
> 2 If the servo-off signal is turned on, alarm DS1933 "NEED REF RETURN (SYNC:MIX:OVL)" will be generated in superimposed control.
> After canceling alarm, perform a manual reference position return.

#2 SVF When an axis under composite control is placed in the servo-off state:
0: Composite control is canceled.
1: Composite control is not canceled.

Follow-up specification follows the setting of bit 0 (FUPx) of parameter No. 1819.
When bit 2 (SVF) of parameter No. 8168 is set to 1, bit 0 (NMR) of parameter No. 8161 is invalid. Bit 1 (MSO) of parameter No. 8168, specification for servo-off, is also invalid.

> **NOTE**
> If a composite control axis is placed in the servo-off state when stopped, set this parameter to 1.

#4 MWR When the synchronous control, composite control, or superimposed control is started or ended and the automatic setting of a workpiece coordinate system is executed, the tool offset number is:
0: Canceled.
1: Not canceled.

> **NOTE**
> This parameter is valid when the workpiece coordinate system (bit 0 (NWZ) of parameter No.8136 is set to 0) is enabled.

#6 WST When a workpiece coordinate system is automatically set up for a slave axis at the end of synchronous control, workpiece coordinate system presetting is:
0: Not performed.
1: Performed.

> **NOTE**
> This parameter is valid when the workpiece coordinate system (bit 0 (NWZ) of parameter No.8136 is set to 0) is enabled, and bit 6 (SPV) of parameter No. 8167 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
8169		SESx				MRFx	MVMx	MDMx

[Input type] Parameter input
[Data type] Bit axis

#0 MDMx As machine coordinates in composite control:
0: Coordinates for the local path are displayed.
1: Coordinates for the other path in composite control are displayed.

#1 MVMx In composite control, machine coordinates (#5021 and above) to be read are:
0: Machine coordinates of the local path.
1: Machine coordinates of the other path in composite control.

#2 MRFx In composite control, the rapid traverse rate is:
0: The rapid traverse rate for the specified axis.
1: The rapid traverse rate for the axis along which a movement is made.

#6 SESx If a synchronization error is out of the tolerable range (specified with parameter No. 8181):
0: Alarm SV0407, “EXCESS ERROR”, is issued.
1: No alarm is issued. Instead, the excess synchronization error signal SEO<Fn559> is output.

SESx is valid when bit 1 (SERx) of parameter No. 8162 is 1. Specify the value of this parameter for the slave axis.

8180	Master axis with which an axis is synchronized under synchronous control

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 101, 102, 103, . . . , (path number)*100+(intra-path relative axis number) (101, 102, 103, . . . , 201, 202, 203, . . . , 1001, 1002, 1003, . . .)
This parameter sets the path number and intra-path relative axis number of the master axis with which each axis is synchronized. When zero is specified, the axis does not become a slave axis and is not synchronized with another axis. When an identical number is specified in two or more parameters, one master axis has two or more slave axes.

8181	Synchronization error limit of each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
When the synchronization deviation detected (bit 1 (SERx) of parameter No. 8162 is set to 1), this parameter specifies the limit of the difference between the positioning deviation of the slave axis and that of the master axis. Set this parameter to the slave axis.

8183	Composite control axis of the other path in composite control for each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 101, 102, 103, . . . , (path number)*100+(intra-path relative axis number) (101, 102, 103, . . . , 201, 202, 203, . . . , 1001, 1002, 1003, . . .)
This parameter sets with which axis each axis is to be placed under composite control. When zero is specified, control of the axis is not replaced under composite control. An identical number can be specified in two or more parameters, but composite control cannot be exercised for all of them at a time.

NOTE
When the two-path interface is used (bit 1 (MIX) of parameter No. 8166 is set to 1), set this parameter for path 2.

8184	Coordinates of the reference point of an axis on the coordinate system of another axis under composite control

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter specifies the coordinates of the reference point of an axis on the coordinate system of another axis under composite control.

> **NOTE**
> This parameter is valid when bit 4 (MPMx) of parameter No. 8162 is set to 1, and workpiece coordinate systems (G54 to G59, including additional workpiece coordinate systems) are not used.

8185	Workpiece coordinates on each axis at the reference position

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the workpiece coordinates on each master axis, subject to synchronous control, when the master and slave axes are at the reference position. This parameter is enabled when bit 1 (SPMx) of parameter No. 8163 is set to 1. Set this parameter for the master axis.

	#7	#6	#5	#4	#3	#2	#1	#0
11284						NCA		

[Input type] Parameter input
[Data type] Bit

#2 NCA When the inside of the specified area on stored stroke check 2,3 is defined as the forbidden area, the combination of axes that composes the forbidden area on Composite control mode is:

0: Exchanged according to the exchanging of axes on Composite control. (Conventional specification)
1: Not exchanged. (Compatible specification with FS0*i*-TTC)

Example) In the following machine compositions,

Path 1	**Path 2**
- X1 axis	- X2 axis
- Z1 axis (Master axis of Composite control)	- Z2 axis (Slave axis of Composite control)

1. In case of bit 2 (NCA) of parameter No.11284 is set to 0 (Conventional specification)

The forbidden area is composed from the axes actually controlled according to the exchanging of Z axes on Composite control, as shown in Fig. 8.6.2 (e).

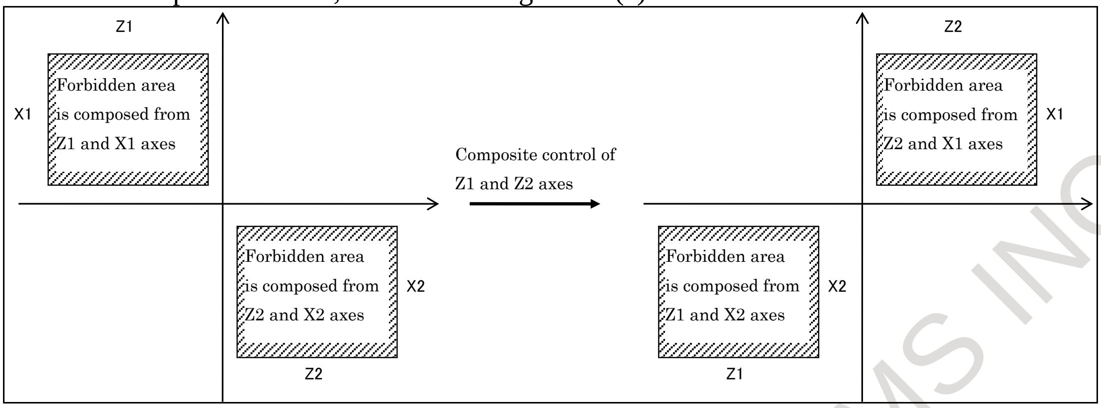


Fig. 8.6.2 (e) Composite control and the inside forbidden area (Existing specification)

2. In case of bit 2 (NCA) of parameter No.11284 is set to 1 (Compatible specification with FS0*i*-TTC)

The forbidden area is kept even if Composite control is executed, as shown in Fig. 8.6.2 (f).

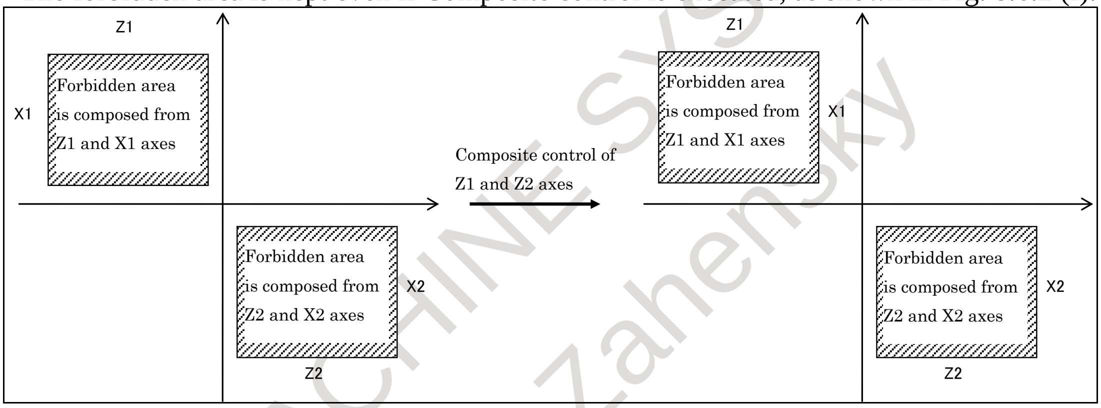


Fig. 8.6.2 (f) Composite control and the inside forbidden area (Compatible specification with FS0*i*-TTC's)

Diagnosis data

The diagnosis data displays synchronous errors.

3502	Display of synchronous error for each axis

[Unit of data] Detection unit

[Meaning] Displays the difference between the master and slave axes in position deviation if synchronous deviation detection is performed (bit 1 (SERx) of parameter No. 8162 =1).

The positional deviation difference is:

(positional deviation of the master axis) ± (positional deviation of the slave axis)

+ if a mirror image is applied to the synchronization command.
- if no mirror image is applied to the synchronization command.

Alarm and message

If any of the following alarms occurs, synchronous or composite control will be canceled for all axes.

- P/S alarm

Number	Message	Description
PS0350	PARAMETER OF THE INDEX OF THE SYNCHRONOUS CONTROL AXIS SET ERROR.	An illegal synchronization control axis number (parameter No. 8180) is set.
PS0351	BECAUSE THE AXIS IS MOVING, THE SYNC CONTROL IS CAN'T BE USED.	While the axis being subject to synchronization control was moving, an attempt was made to start or cancel the synchronization control by a synchronization control axis selection signal.
PS0352	SYNCHRONOUS CONTROL AXIS COMPOSITION ERROR.	This error occurred when: 1) An attempt was made to perform synchronization control for the axis during a synchronization, composition, or superposition. 2) An attempt was made to synchronize a further great-grandchild for a parent-child-grandchild relation. 3) An attempt was made to operate synchronization control although a parent-child-grandchild relation was not set.
PS0353	THE INSTRUCTION WAS DONE FOR THE AXIS WHICH WAS NOT ABLE TO MOVE.	This error occurred when: - For synchronization 1) A move command was issued to the axis for which bit 7 (NUMx) of parameter No. 8163 is set to 1. 2) A move command was issued to the slave axis. - For composition 1) A move command was issued to the axis for which bit 7 (NUMx) of parameter No. 8163 is set to 1. 2) A move command was issued to the axis for which bit 7 (MUMx) of parameter No. 8162 is set to 1.
PS0354	THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN SYNC MODE	This error occurred when G28 was specified to the master axis being parking during synchronization control, but an axis reference position is not set for the slave axis.
PS0355	PARAMETER OF THE INDEX OF THE COMPOSITE CONTROL AXIS SET ERROR.	An illegal composite control axis number (parameter No. 8183) is specified.
PS0356	BECAUSE THE AXIS IS MOVING, THE COMP CONTROL IS CAN'T BE USED.	While the axis being subject to composite control was moving, an attempt was made to start or cancel the composite control by a composite control axis selection signal.
PS0357	COMPOSITE CONTROL AXIS COMPOSITION ERROR.	This error occurred when an attempt was made to perform composite control for the axis during a synchronization, composition, or superposition.
PS0359	THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN COMP MODE	This error occurred when G28 was specified to the composite axis during composite control, but a reference position is not set to the other part of the composition.
PS0367	3-D CONV. WAS COMMANDED IN SYNC MODE AS THE PARAMETER PKUx(NO.8162#2) IS 0.	A 3-dimensional coordinate conversion was commanded during synchronization control when the bit 2 (PKUx) of parameter No. 8162 was 0.

- D/S alarm

Number	Message	Description
DS1933	NEED REF RETURN(SYNC:MIX:OVL)	The relation between a machine coordinate of an axis in synchronization, composition, or superposition control, and the absolute, or relative coordinate was displaced. Perform the manual return to the reference position.

- Servo alarm

Number	Message	Description
SV0407	EXCESS ERROR	The difference value of the amount of positional deviation for the synchronization axis exceeded the setting value. (during synchronization control only)

Caution

- Items common to synchronous control and composite control

⚠ **CAUTION**

1 At the start or end of synchronous or composite control, the tool must be stopped on the axis subject to that control.

2 Before or after an M code for starting or canceling synchronous or composite control during automatic operation, be sure to specify a waiting M code (M code without buffering). When exercising synchronous or composite control in a path, be sure to prohibit look-ahead operation by specifying an M code without buffering before or after an M code for starting or canceling synchronous or composite control.

3 The axes under synchronous or composite control must match in least command increment, Detection unit, and diameter/radius specification. Otherwise, the amounts of travel will not be equal.

4 In synchronous or composite control, do not change the parameters related to it (including axis control, increment system, feedrate, and acceleration/deceleration control).

5 Before starting synchronous or composite control, make sure that the axes subject to it has undergone a reference position return after the power was turned on or that a reference position has been established with an absolute pulse coder.

6 If an emergency stop or servo off is performed or if a servo alarm occurs, the coordinates will change due to synchronous or composite control. If this occurs alarm DS1933, “NEED REF RETURN (SYNC:MIX:OVL)" will be generated. After canceling the emergency stop, canceling servo off, or restoring from the servo alarm, perform a reference position return and coordinate system setting first, then start synchronous or composite control.

7 Acceleration/deceleration control, pitch error compensation, backlash compensation, and stored stroke check are performed independently, regardless of synchronous or composite control.

8 The following servo software functions cannot be used with synchronous or composite control:
- Abnormal load detection and switching function for each of cutting and rapid traverse
- Fine acceleration/deceleration and switching function for each of cutting and rapid traverse

9 The following functions cannot be used in synchronous or composite control:
- Electronic gear box
- Spindle positioning
- Manual numeric command

10 During synchronous or composite control, reference position establishment based on "linear scale with absolute address reference mark" or "linear scale with absolute address zero point" is impossible.

NOTE
1. This function is an option function.
2. You can place more than one axis under synchronous or composite control at the same time. You cannot, however, synchronize an axis to multiple axes at the same time, synchronize the interchanging axis under composite control to another axis, or duplicate an interchange.
3. Synchronous or composite control cannot be performed on a linear axis and a rotation axis.

- **Items related to synchronous control only**

⚠ **CAUTION**
1. In synchronous control, you cannot issue a move command for the synchronous axis on the synchronous slave.
2. The axes under synchronous control must match in acceleration/deceleration time constant and servo parameters, insofar as possible. If their settings greatly differ, the actual machine movement will deviate.
3. Even if you perform operations that do not move the machine but change the workpiece coordinate system only, such as workpiece coordinate system setting/shifting and geometric offset commands, on the synchronous master, this will not be reflected in the workpiece coordinate system on the slave.
4. If you perform workpiece coordinate system setting/wear offset commands, tool nose radius compensation, or cutter compensation on the synchronous master, the travel path on the synchronous slave will shift by the offset, but the offset will not be set as an offset amount. (No offset vector will be created.)
5. The synchronous or composite control and axis synchronous control can be specified at the same time. In this case, the master axis subject to axis synchronous control can be set as the master axis subject to synchronous control. The master axis subject to axis synchronous control cannot be used as the slave axis subject to synchronous control, and the slave axis subject to axis synchronous control cannot be used as the master or slave axis subject to synchronous control.
6. If you want to perform a tool retract and recover operation for axes in the synchronous control mode, perform the operation for the master axis. In the same way as in ordinary synchronous control, the master and slave axes move together.

- **Items related to composite control only**

⚠ **CAUTION**
1. If you want to place a tapping axis under composite control, place it under composite control first, and then issue a rigid tapping command. In rigid tapping mode, you must not switch the composite control axis selection signal; be sure to do this in the rigid tapping canceled state.
2. If you place a tapping axis under composite control, rigid tapping will be performed using the time constant, loop gain, in-position width, and the positional deviation limits during stoppage/travel of the interchanging axis in composite control.

> ⚠ **CAUTION**
> 3 In composite control, the loop gain may be switched depending on the switching of the Cs contour control switching signal and of the composite control axis selection signal. For this reason, switch the signals in the state in which the tool has stopped on the related axis. If you change the loop gain while the tool is moving along an axis, shock may result.
> 4 The synchronous or composite control and axis synchronous controlcan be specified at the same time. An axis used for movement under composite control can be selected as the master axis for axis synchronous control, but cannot be specified as a slave axis for axis synchronous control.
> 5 If you want to perform a tool retract and recovery operation for axes in the composite control mode, perform the operation for the composite control source axis. In the same way as in ordinary composite control, the composite control destination axis moves.

Limitation

- Limitations on synchronous control and composite control

Function	In synchronous control	In composite control
Acceleration/deceleration control	The synchronous slave axis is subject to acceleration/deceleration of the same type as the synchronous master axis. As the time constants, those specific to the axes are used.	The acceleration/deceleration type of the specified path is used. As the time constants, those specific to the axes are used (3).
Look-ahead acceleration/deceleration before interpolation	The synchronous slave axis is subject to the same acceleration/deceleration as the synchronous master axis.	The acceleration/deceleration settings of the specified path are used.
Cutting feedrate clamp	Clamping is performed on the synchronous master.	Clamping is performed on the specified path (4).
Reference position return	Possible on the synchronous master axis if the axis is not in parking. Only automatic reference position return (G28) is possible if the synchronous master axis is in parking.	Possible on axes not related to composite control. Only automatic reference position return (G28) is possible on an axis under composite control.
2nd, 3rd, 4th reference position return	Same as above.	Same as above.
Reference position return check	Same as above.	Same as above.
PMC axis control	Possible on axes other than the synchronous slave axis. (If a slave axis is specified as a PMC control axis, alarm PS0130 is issued.)	Possible.
Polar coordinate interpolation (T series), Cylindrical interpolation	Possible.	Possible. (In polar coordinate interpolation (T series) or cylindrical interpolation mode, composite control cannot be turned ON/OFF.)
Handle interruption	See bit 5 (SMI) of parameter No. 8163.	Possible with bit 6 (MMI) of parameter No. 8163
Mirror image	See bit 5 (SMI) of parameter No. 8163.	Signals in the specified path are effective (4).
Machine lock	Signals in the individual paths are effective (1).	Signals in the specified path are effective (4).
Inter lock	Signals on the synchronous master side are valid for the synchronous slave axis.(2)	Signals in the specified path are effective (4).

Function	In synchronous control	In composite control
Override	Signals on the synchronous master side are valid for the synchronous slave axis.(2)	Signals in the specified path are effective (4).
External deceleration	Signals on the synchronous master side are valid for the synchronous slave axis.(2)	Signals in the specified path are effective (4).
Skip function	Not possible on the synchronous slave axis.	Possible on axes not related to composite control.
Automatic tool offset	Not possible on the synchronous slave axis.	Possible on axes not related to composite control.
Follow-up	Not possible during synchronization.	Not possible during composite control.
Program restart	Not possible with a program containing synchronous control.	Not possible with a program containing composite control.
Cs contour control	Synchronous control possible (5).	Composite control possible (5).
Spindle positioning	Synchronous control does not possible	Composite control does not possible
EGB function	Synchronous control does not possible	Composite control does not possible
Servo off	Synchronous control does not possible	Possible unless the interchanging axis in composite control is in the servo off state.

(1) Processed after synchronous pulses are transferred to the slave.
(2) Processed on the master, then transfers synchronous pulses.
(3) Transfers composite control pulses and the acceleration/ deceleration type. As the time constant, uses that on the slave.
(4) Processed on the master, then transfers composite control pulses.
(5) Limited to combinations of Cs axes.

- Reading of coordinates in synchronous or composite control

In synchronous or composite control, the reading of custom macro system variable positional information or of current coordinates from the PMC window is as follows:

Positional information type	In synchronous control	In composite control
Absolute coordinate	Reading possible.	Reading possible.
Machine coordinate	Reading possible.	Reading possible.
End point of each block	Reading possible on the master only.	Reading possible.
Skip signal position	Reading possible on the master only.	Reading not possible.

- Canceling synchronous or composite control

Synchronous or composite control is canceled in the event of the following, as well as when the synchronous/composite control axis selection signal becomes 0.
(1) Emergency stop
(2) Reset
(3) Servo alarm
(4) Servo off *1
(5) Overtravel *2
(6) Alarm related to synchronous or composite control
(7) Alarm PW0000

If one of the above events occurs in either path, all paths will be released from synchronous or composite control. If one of the above events occurs in either path in synchronous or composite control, the other path will automatically be placed in feedhold state (during automatic operation) or in interlock state (during manual operation).

*1: By setting bit 0 (NMR) of parameter No. 8161 to 1, you can prohibit the composite control axis from release from the synchronous or composite control state even if the axis enters the servo off state.

By setting bit 7 (NSR) of parameter No. 8161 to 1, you can prohibit the synchronous control axis from release from the synchronous or composite control even if the axis enters the servo off state.

*2: By setting bit 5 (NCS) of parameter No. 8160 to 1, you can prohibit the synchronous or composite control axis from release from the synchronous or composite control state even if the axis enters the overtravel state.

- Axis state output signals in synchronous or composite control

State output signal	In synchronous control	In composite control
Axis moving signals MVn <Fn102>	- During movement along the master or slave axis, the master axis signal goes 1. - The slave axis signal is always 0 (1).	- On the axis on which a move command is executed, the signal goes 1. The signal on the axis along which the tool is actually moving does not turn to 1 (1).
Axis moving direction signals MVDn <Fn106>	- On the master axis, the moving direction of the master axis. - On the slave axis, moving direction after mirror image processing.	- Actual moving direction on the axis (moving direction after mirror image processing in composite control).
In-position signals INPn <Fn104>	- The master axis signal goes 1 when both the master and slave axes are in the in-position state. - The slave axis signal is always 1.	- The signal in the specified path reflects the state of the moving axis with that specification.
Reference position establishment signal ZRFn <Fn120>	- The signal on the axis for which the reference position has been established turns to 1.	- The signal on the axis for which the reference position has been established turns to 1.
Reference position return completion signal ZPn <Fn094>	- After completion of reference position return of the master axis, the signal on the master axis turns to 1. - The slave axis is synchronized with movement of the master axis, but is not subjected to reference position return. The signal turns to 0. - When the master axis is parking, the slave axis is subjected to reference position return. The signal on the slave axis turns to 1.	- The signal on the axis along which the tool is actually moving turns to 1.

(1) In a positional deviation check, regardless of the states of these signals, parameter No. 1828 is used if move command pulses are sent to the motor (regardless of whether the axis is the master or slave) as the limit, and parameter No. 1829 if none are sent.

Examples of Use

- Examples of independent control and of synchronous control on the Z1 and Z2 axes

(1) Machine configuration

(a) Independent control

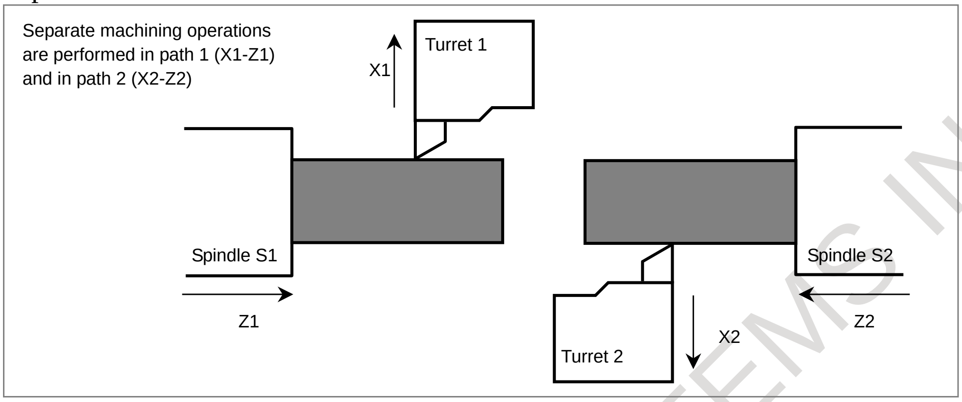


Fig. 8.6.2 (g)

(b) Z1-Z2 axis synchronous control

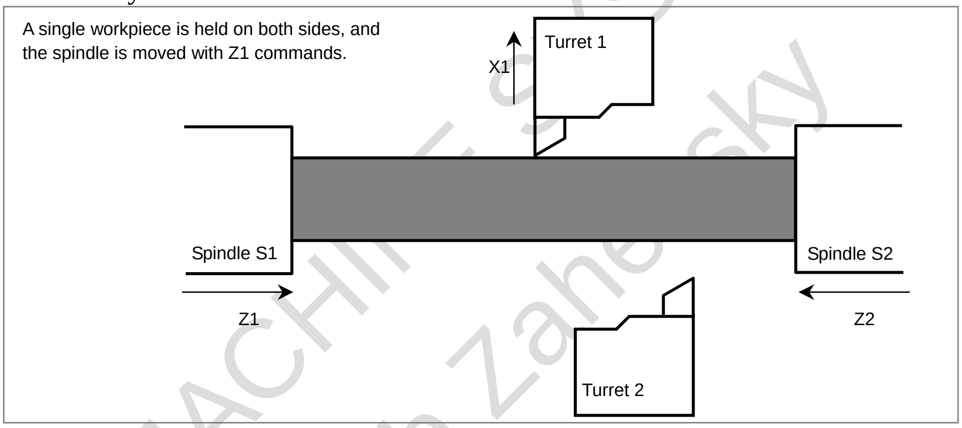


Fig. 8.6.2 (h)

(2) Parameter settings

- To synchronize the Z2 (slave) axis with the Z1 (master) axis, set parameter No. 8180z of path 2 to "102".
- Because the plus directions of the Z1 (master) and Z2 (slave) axes are opposite, perform synchronization with a mirror image applied. To that end, set bit 0 (SMRz) of parameter No. 8162 of path 2 to 1.
- Because the movements along the Z1 (master) and Z2 (slave) axes must be equal, perform synchronous error detection. Set bit 1 (SERz) of parameter No. 8162 of path 2 to 1. Set the synchronous error limit in parameter No. 8181z of path 2, in the range of 100 to 1000. (The setting differs with each machine.)
- During synchronization, the difference between the Z1 (master) axis and the Z2 (slave) axis in positional deviation is displayed as No. 3502 on the diagnosis data of path 2.

(3) Signal operation

- To start synchronous movement along the Z1 (master) axis and the Z2 (slave) axis, set signal SYNC2 <G1138.1> to 1.
- To cancel synchronization, set signal SYNC2 <G1138.1> to 0.
- When an emergency stop, NC reset, or alarm occurs, set signal SYNC2 <G1138.1> to 0.
- Set signals other than signal SYNC2 <G1138.1> to 0.

(4) Program example

Path 1		Path 2		
N1000	. . .	N2000	. . .	Independent machining of each path
N1010	Z80.0 ;	N2010	Z150.0 ;	Move the workpiece and the chuck to position
N1020	M200 P12 ;	N2020	M200 P12 ;	Waiting
		N2030	M61 ;	Workpiece clamping and start of synchronization
N1040	M201 P12 ;	N2040	M201 P12 ;	Waiting
N1050	M3 S800 ;			Forward spindle rotation
N1060	Z- 25.0 ;			Movement along the Z1 axis
N1070	. . .			Machining along the X1 and Z1 axes
N1080	M200 P12 ;	N2080	M200 P12 ;	Waiting
		N1090	M62 ;	Cancellation of synchronization and unclamping of workpiece
N1100	M201 P12;	N2100	M201 P12;	Waiting
N1110	. . .	N2110	. . .	Independent machining of each path

where
M61 is an M code that clamps the workpiece and sets signal SYNC2 <G1138.1> to 1, and
M62 is an M code that sets signal SYNC2 <G1138.1> to 0 and unclamps the workpiece.

NOTE

1. An operation to make the speeds of spindles S1 and S2 identical is required. To do this, issue a spindle command of path 1 to both S1 and S2.
2. A waiting M code is an M code without buffering.

- Examples of independent control and interpolation on the X1 and Z2 axes

(1) Machine configuration

(a) Independent control

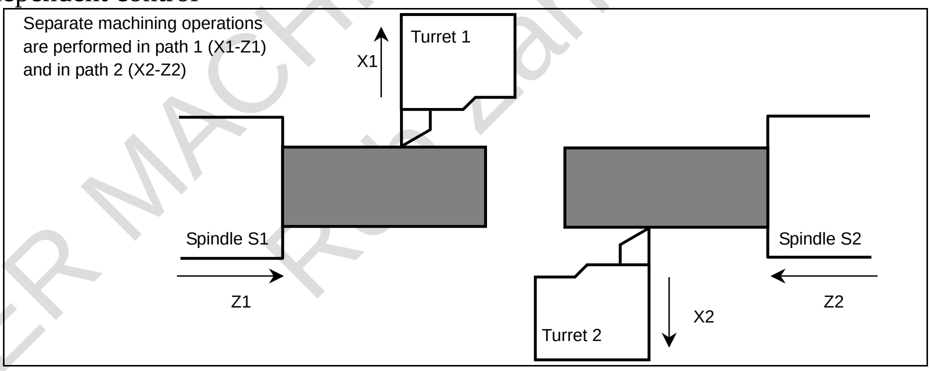


Fig. 8.6.2 (i)

(b) Interpolation on the X1 and Z2 axes

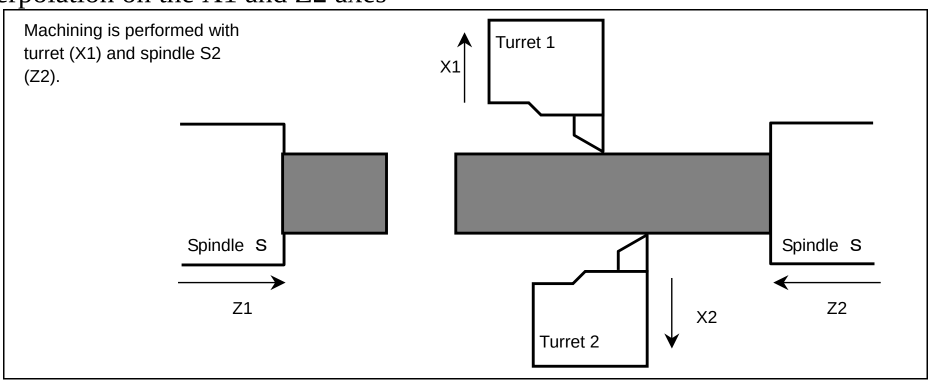


Fig. 8.6.2 (j)

There are two ways to perform interpolation on the X1 and Z2 axes.

1. Use the program of path 2 to issue a command to the X2 and Z2 axes to synchronize the X1 (slave) axis to the X2 (master) axis and to park the X2 (master) axis. For path 1, do not issue a move command.
2. Perform composite control to let the X1 and X2 axes interchange between paths. For path 1, do not issue a move command.

The following explains the ways using synchronous control and composite control separately.

If using synchronous control

(1) Parameter settings

- To synchronize the X1 (slave) axis to the X2 (master) axis, set parameter No. 8180x of path 1 to "201".
- For both X1 (slave) and X2 (master) axes, no mirror image is required because the direction away from the center of the workpiece is the plus direction of coordinates.
- Do not perform synchronous error detection because the X2 (master) axis is parked.
- During synchronization, the difference between the X2 (master) and X1 (slave) axes in positional deviation is displayed as No. 3502 on the diagnosis data of path 1.

(2) Signal operation

- To start synchronous control on the X2 (master) axis and the X1 (slave) axis, set signals SYNC1 <G0138.0> and PK1 <G1122.0> to 1.
- To cancel synchronization, set signals SYNC1 <G0138.0> and PK1 <G1122.0> to 0.
- When an emergency stop, NC reset, or alarm occurs, set SYNC1 <G0138.0> and PK1 <G1122.0> to 0.
- Set signals other than SYNC1 <G0138.0> and PK1 <G1122.0> to 0.

(3) Program example

Path 1		Path 2		
N1000	. . .	N2000	. . .	Independent machining of each path
N1010	Z0 ;	N2010	Z20.0 ;	Move each workpiece to position
N1020	X120.0 ;	N2020	X120.0 ;	Move each X axis to the synchronization start position (X1=X2)
N1030	M200 P12;	N2030	M200 P12 ;	Waiting
		N2040	M55 ;	Synchronize the X2 and X1 axes and start parking on the X2 axis
N1050	M201 P12 ;	N2050	M201 P12 ;	Waiting
		N2060	T0212 ;	Set the offset for turret 1
		N2070	S1000 M4 ;	Reverse spindle rotation
		N2080	G0 X30.0 Z55.0 ;	
		N2090	G1 F0.2 W-15.0 ;	Machining along the X1 and Z2 axes
		N2100	. . .	
N1110	M200 P12 ;	N2110	M200 P12 ;	Waiting
		N2120	M56 ;	Cancellation of synchronization and parking
N1130	M201 P12 ;	N2130	M201 P12 ;	Waiting
N1140	. . .	N2140	. . .	Independent machining of each path

where

M55 is an M code to start the control of turret 1 with the program of path 2, and M56 is an M code that cancel the control of turret 1 with the program of path 2.

> **NOTE**
> During synchronous control on the X axis, no move command can be issued from path 1 to the X1 (slave) axis, but movement along the Z1 axis is possible.

If using composite control

(1) Parameter settings

- To perform composite control by letting the X1 and X2 axes interchange, set parameter No. 8183x of path 2 to "101".
- Because the direction of the coordinates on the X1 axis is opposite from that of the coordinates on the X2 axis, set bit 6 (MCDx) of parameter No. 8162 of path 2 to 1.
- To automatically set the position of turret 1 in the workpiece coordinate system of path 2 at the start of composite control, set bit 4 (MPMx) of parameter No. 8162 of path 2 to 1.
- To automatically set the position of turret 1 in the workpiece coordinate system of path 1 at the end of composite control, set bit 5 (MPSx) of parameter No. 8162 of path 1 to 1.
- Assuming that the X coordinate of the reference position of turret 1 in the workpiece coordinate system of path 2 is -150.0 mm as shown in the figure below, set parameter No. 8184x of path 2 to "-150000" to automatically set the coordinate system.

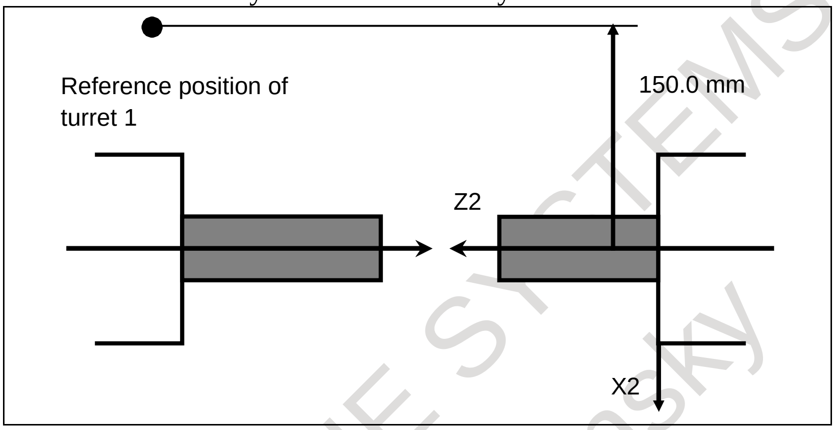


Fig. 8.6.2 (k)

(2) Signal operation

- To start composite control on the X2 and X1 axes, set signal MIX1 <G1128.0> to 1.
- To cancel composite control, set signal MIX1 <G1128.0> to 0.
- When an emergency stop, NC reset, or alarm occurs, set MIX1 <G1128.0> to 0.
- Set signals other than MIX1 <G1128.0> to 0.

(3) Program example

Path 1		Path 2		
N1000	. . .	N2000	. . .	Independent machining of each path
N1010	Z0 ;	N2010	Z20.0 ;	Move each workpiece to position
		N2020	X120. ;	Move along the X2 axis to a non-interference position
N1030	M200 P12 ;	N2030	M200 P12 ;	Waiting
		N2040	M55 ;	Start composite control on the X2 and X1 axes (perform automatic coordinate system setup)
N1050	M201 P12 ;	N2050	M201 P12 ;	Waiting
		N2060	T0212 ;	Set the offset for turret 1
		N2070	S1000 M4 ;	
		N2080	G0 U10.0 W- 20.0 ;	Machining along the X1 and Z2 axes
		N2090	G1 F0.2 W- 15.0 ;	
		N2100	. . .	
N1110	M200 P12 ;	N2110	M200 P12 ;	Waiting
		N2120	M56 ;	Cancel composite control (perform automatic coordinate system setup)
N1120	M201 P12 ;	N2120	M201 P12 ;	Waiting
N1130	. . .	N2130	. . .	Independent machining of each path

where
M55 is an M code to start the control of turret 1 with the program of path 2, and
M56 is an M code that cancel the control of turret 1 with the program of path 2.

NOTE

1 At the start and end of composite control, you do not necessarily perform automatic setup of the coordinate system. If you do not perform automatic setting, the program will set an appropriate one internally.
2 During composite control on the X axis, you can issue a move command for the X axis in path 1 to move the tool along the X2 axis.
3 With the parameter settings above, turret 1 will be positioned on the minus side of the X coordinates in the workpiece coordinate system of path 2. For this reason, specify with the opposite sign as usual, such as U+10.0 and U-10.0 to move turret 1 toward the center of the workpiece and move it away from it, respectively.
   If this is inconvenient, specify the following parameter settings:
   Bit 6 (MCDx) of parameter No. 8162 = 0
   Parameter No. 8184x = 150000
   These settings cause turret 1 to exist on the plus side of the X coordinates virtually.

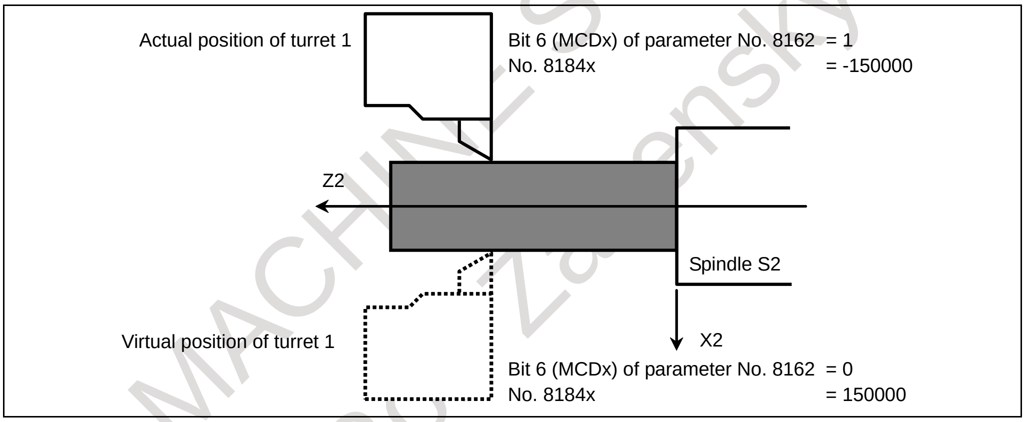


Fig. 8.6.2 (l)

- Examples of independent control and of interpolation on X1-Z2 and X2-Z1

(1) Machine configuration

(a) Independent control

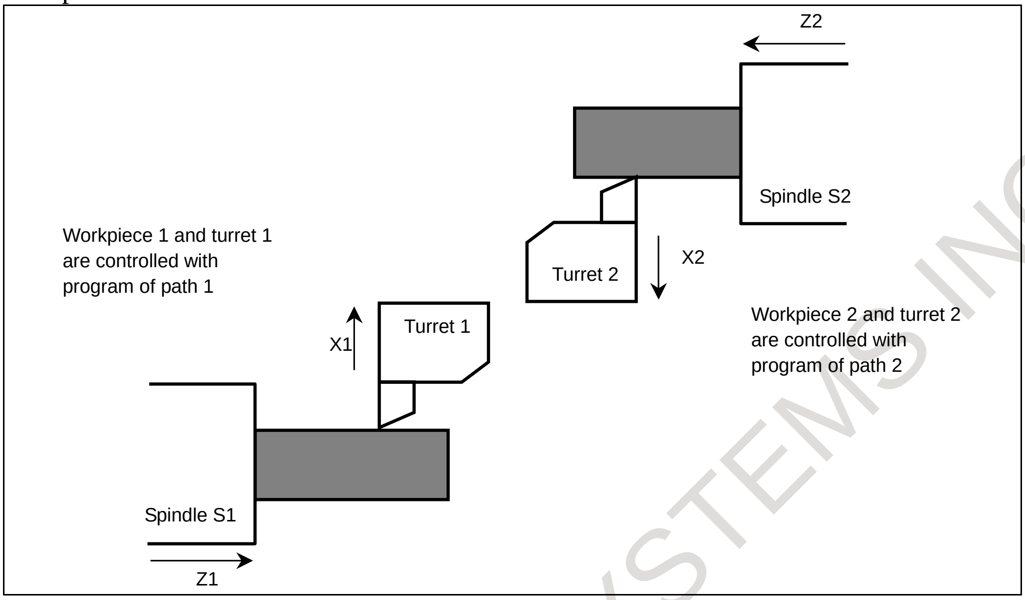


Fig. 8.6.2 (m)

(b) X1-Z2 and X2-Z1 interpolation

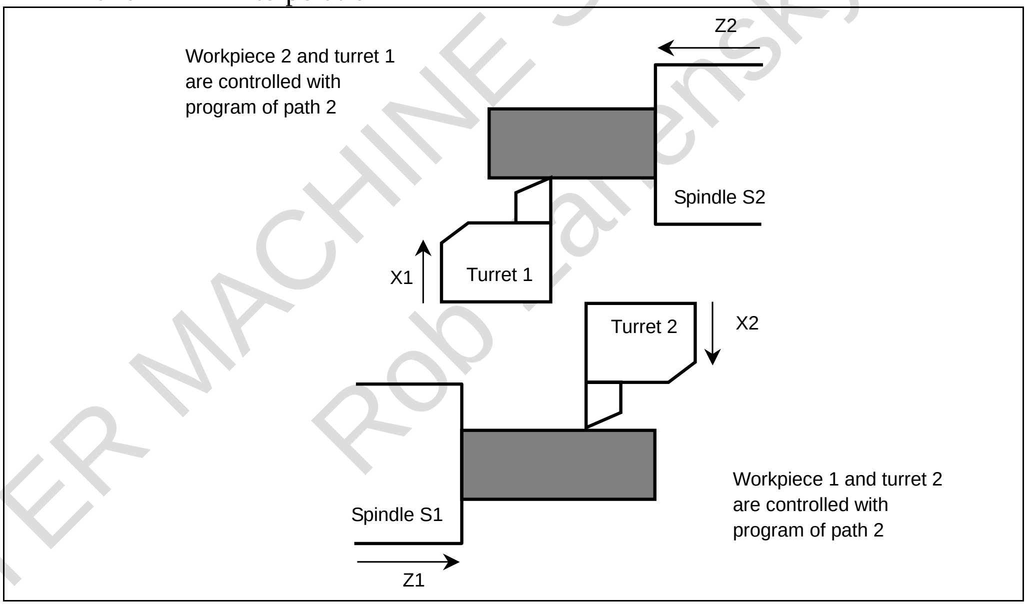


Fig. 8.6.2 (n)

(2) Parameter settings

- To perform composite control by letting the X1 and X2 axes interchange, set parameter No. 8183x of path 2 to "101".
- Because the direction of the coordinates on the X1 axis is opposite from that of the coordinates on the X2 axis, set bit 6 (MCDx) of parameter No. 8162 of path 2 to 1.
- To automatically set the position of the remote turret in the workpiece coordinate system of each path at the start of composite control, set bit 4 (MPMx) of parameter No. 8162 of each of paths 1 and 2 to 1.
- To automatically set the position of the local turret in the workpiece coordinate system of each path at the end of composite control, set bit 5 (MPSx) of parameter No. 8162 of each of paths 1 and 2 to 1.

- Assuming that the relations between the workpiece coordinates of each path and the reference position are as shown in the figure below, set parameter No. 8184x of path 1 to "200000" and No. 8184x of path 2 to "180000".

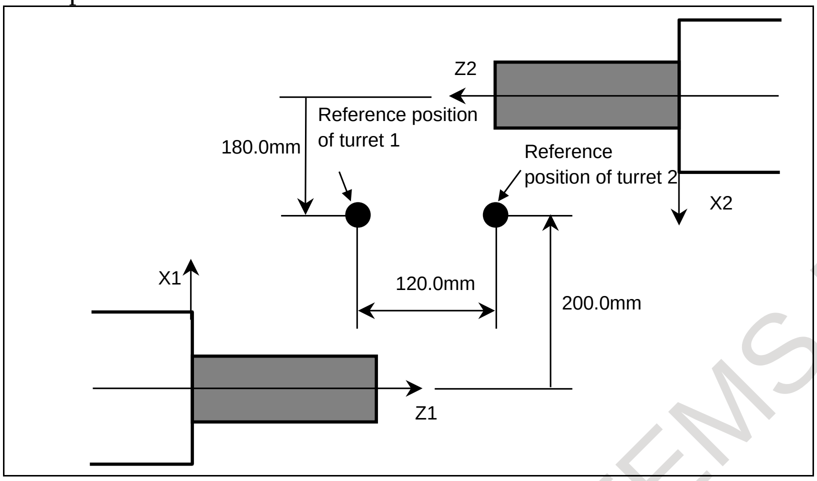


Fig. 8.6.2 (o)

(3) Signal operation

- To start composite control on the X2 and X1 axes, set signal MIX1 <G1128.0> to 1.
- To cancel composite control, set signal MIX1 <G1128.0> to 0.
- When an emergency stop, NC reset, or alarm occurs, set signal MIX1 <G1128.0> to 0.
- Set signals other than signal MIX1 <G1128.0> to 0.

(4) Program example

Path 1	Path 2	
N1000 . . .	N2000 . . .	Independent machining of each path
N1010 M200 P12 ;	N2010 M200 P12 ;	Waiting
	N2020 M55 ;	Start composite control on the X1 and X2 axes
N1030 M201 P12 ;	N2030 M201 P12 ;	Waiting
N1040 T0313 ;	N2040 T0212 ;	Selection of composite control tool and setting of offset
N1050 G50 W120.0 ;	N2050 G50 W120.0 ;	Shifting of Z axis workpiece coordinate system
N1060 S1000 M4 ;	N2060 S1500 M4 ;	
N1070 G0 X20.0 Z15.0 ;	N2070 G0 X15.0 Z30.0;	Start of composite control
N1080 G1 F0.5 W- 8.0 ;	N2080 G1 F0.1 W-5.0 ;	
N1090 . . .	N2090 . . .	
N1100 M200 P12 ;	N2100 M200 P12 ;	Waiting
	N2110 M56 ;	End of composite control
N1120 M201 P12 ;	N2120 M201 P12 ;	Waiting
N1130 G50 W- 120.0 ;	N2130 G50 W-120.0 ;	Shifting of Z axis workpiece coordinate system
N1140 . . .	N2140 . . .	Independent machining of each path

where

M55 is an M code to start composite control (set signal MIX1 <G1128.0> to 1), and
M56 is an M code to cancel composite control (set signal MIX1 <G1128.0> to 0).

> **NOTE**
> At the start and end of composite control, you do not necessarily perform automatic setup of the coordinate system. If you do not perform automatic setting, the program will set an appropriate one internally.

Others

- In the preceding explanation, a single synchronous control or composite control operation is explained. In reality, however, you can select multiple synchronous control or composite control operations and execute them at the same time: Set all necessary parameters in advance, and specify which synchronous or composite control to perform, using appropriate signals. Make sure that a single axis is not subject to multiple synchronous/composite control operations at the same time.
- You can set only a single set of axes subject to synchronous or composite control. If you require multiple sets, use the programmable parameter input function (G10) to change the parameter settings in the program. Before changing parameters, make sure that the associated synchronous or composite control has been canceled. If parameters are changed during Synchronous control or Composite control, the changes will not take effect immediately. For the changes made to the parameters to take effect, it is necessary to turn off synchronous or composite control mode and turn it back on.

  (Example)

  Change parameter settings to synchronize the Z2 (slave) axis to the Y1 (master) axis.
  (To set parameter No. 8180z of path 2 to "103", execute a program such as that shown below.)

  ```
  N0200   . . .
  N0210   G10   L50   ;              Start of parameter settings
  N0220   N8180   P2   R103   ;      Set parameter No. 8180z to "103".
  N0230   G11   ;                    End of parameter settings
  N0240   . . .
  ```

  Execute blocks G10 and G11 while the Y1 and Z2 axes are not subject to synchronous or composite control.

Troubleshooting

1. Unable to start synchronous or composite control. (No alarm occurs.)
   (1) The correct synchronous or composite control have not been set up.
   → To perform synchronous or composite control, you must set up the correct.
   (2) The synchronous/composite control axis selection signals <Gn128> or <Gn138> is not changed from 0 to 1.
   → Synchronous or composite control starts when the synchronous/composite control axis selection signal <Gn128> or <Gn138> is changed from 0 to 1. After synchronous or composite control ends due to a reset or alarm, merely canceling the reset or alarm cause will not cause synchronous or composite control to restart. After canceling the reset or alarm cause, you must make the signal rise again.
   (3) The axis number of the axis subject to synchronous or composite control is not set in the appropriate parameter.
   → To perform synchronous control, set the axis number of the master axis in parameter No. 8180. To perform composite control, set the axis number of the interchanging axis in parameter No. 8183. If using a 2-path interface, use the parameter of path 2.
   (4) Synchronous or composite control cannot be started if the NC unit is one of the following states:
   - Emergency stop
   - Reset
   - Servo alarm
   - Alarm PW0000
   - Alarm related to synchronous or composite control

Synchronous or composite control cannot be started, either, if the axis on which synchronous or composite control is to start is either of the following states:

- Servo off
- Overtravel

2. An attempt to raise a synchronous or composite start signal results in an alarm.
   (1) An attempt is made to use an axis already under synchronous or composite control for another synchronous or composite signal operation. (Alarms PS0352 and PS0357)
      → You cannot use a single axis for a combination of multiple synchronous or composite control operations. Note, however, that the synchronous master axis can be the master of multiple synchronous slave axes at the same time. You can add additional slave axes while synchronous control is in progress.
   (2) The axis number set in a parameter is greater than the number of controllable axes. (Alarms PS0350 and PS0355)
      → The number of the synchronous master axis and that of the interchanging axis subject to composite control must not exceed the number of axes controllable in the remote path (or the local path if you are to perform synchronous control in that path).
   (3) The tool is moving along the axis that you want to control. (Alarms PS0351 and PS0356)
      → At the start of synchronous or composite control, the tool must be stopped on the axis subject to that control. That the tool is stopped on an axis means that the speed specified after acceleration/deceleration is zero.
   (4) An axis on which you want to perform synchronous or composite control belongs to another machine group. (Alarms PS0350 and PS0355)
      → The axes on which you want to perform synchronous or composite control must be of the same machine group.

3. An attempt to cancel synchronous or composite control results in alarms PS0351 and PS0356.
   (1) The tool is moving along the axis on which the control you want to cancel.
      → To cancel synchronous or composite control, the tool must be stopped on the axis on which to cancel it. That the tool is stopped on an axis means that the speed specified after acceleration/deceleration is zero. Before canceling, make sure that the movement in progress signal Fn102 has changed to 0.

4. During synchronous control or composite control, an alarm occurs.
   (1) A move command is issued for a synchronous slave axis. (Alarm PS0353)
      → No move command can be issued for a synchronous slave axis, whether it be automatic or manual.
   (2) A move command is issued for an axis under composite control for which bit 7 (MUMx) of parameter No. 8162 is set to 1. (Alarm PS0353)
      → During composite control, no move command can be issued for an axis for which bit 7 (MUMx) of parameter No. 8162 is set to 1, whether it be automatic or manual.
   (3) During synchronous control, an automatic reference position return command with G28 is issued for a parking master axis. (Alarm PS0354)
      → In this case, a reference position return is performed on the slave axis, requiring that the reference position of the slave axis be established.
   (4) During composite control, an automatic reference position return command with G28 is issued. (Alarm PS0359)
      → In this case, a reference position return is performed on the remote interchanging axis, requiring that the reference position of the interchanging axis be established.
   (5) During synchronous control, a 3-dimensional coordinate conversion is specified. (Alarm PS0367)
      → For the synchronous master axis to be specified in 3-dimensional coordinate conversion mode, set bit 2 (PKUx) of parameter No. 8162 to 1.

5. During synchronous control, servo alarm SV0407 occurs.
   (1) The difference between the synchronous master and slave axes in positional deviation is too large.
   → When the difference between synchronous master and slave axes in positional deviation exceeds the parameter (No.8181) while checking the synchronous error, alarm (SV0407) occurs. In general, this alarm occurs if the difference between the synchronous master and slave axes in acceleration/deceleration time constant or servo parameter is large. It can also occur if the actual movement of the machine is not correct (is not correctly synchronized) due to some cause or other.

6. During synchronous control, the machine position deviates.
   (1) The synchronous master and slave axes in acceleration/deceleration time constant or servo parameter (such as loop gain) differs.
   → In this synchronous control, acceleration/deceleration control and servo control are performed in the master and slave independently of each other. (That is, move commands are synchronized.) For this reason, if the master and slave differ in acceleration/ deceleration time constant and servo characteristics, the actual movement of the machine may not fully match.

7. The travel distance is not correct during synchronous or composite control.
   (1) The master and slave axes differ in diameter/radius specification or inch/millimeter setting.
   → In synchronous or composite control, diameter/radius conversion or inch/millimeter conversion is not performed between master and slave. The master and slave must match in least command increment.

8. After a switch to synchronous or composite control, the tool does not move to the specified coordinates.
   (1) After a switch to synchronous or composite control, a move command is issued without waiting (with an M code without buffering).
   → If, during automatic operation, you want to start or cancel synchronous or composite control with automatic coordinate system setting, be sure to perform waiting (with an M code without buffering) before and after that.

9. Synchronous or composite control is canceled although the synchronous/composite control axis selection signal <Gn128> or <Gn138> is not changed from 1 to 0.
   → In synchronous or composite control, synchronous or composite control will be automatically canceled if at least either of the paths enters one of the following states:
   - Emergency stop
   - Reset
   - Servo alarm
   - Alarm PW0000
   - Alarm related to synchronous or composite control

   In addition, if at least one of the axes on which synchronous or composite control is in progress enters either of the following states:
   - Servo off
   - Overtravel

10. Alarm DS1933 occurs.
   → In synchronous or composite control, alarm DS1933 occurs if at least one path enters one of the following states:
   - Emergency stop
   - Servo off
   - Servo alarm

After canceling the emergency stop, after canceling servo off, or after restoring from the servo alarm, first perform a reference position return or coordinate system setting and then start synchronous or composite control.

8.6.3 Hypothetical Cs Axis Control

Overview

This function allows you to add a hypothetical Cs axis to a path to which no serial spindle is actually connected.
Usually, performing composite control on Cs axes in multi-path control requires that both of the axes to be interchanged between paths be Cs axes. Using this function, however, enables composite control in which a hypothetical Cs axis and a Cs axis to which a serial spindle is actually connected interchange with each other. This makes it possible to move the serial spindle with Cs contour control that is connected to another path with a command to move from a path to which a serial spindle is not connected to a hypothetical Cs axis.

Explanation

By setting bit 7 (CDMx) of parameter No. 1014 to 1 and setting a Cs axis in parameter No. 1023, you can add the hypothetical Cs axis to a path to which a serial spindle is not actually connected.

In multi-path control, this enables composite control in which a hypothetical Cs axis and an actual Cs axis connected to another path interchange with each other.

For the hypothetical Cs axis, you need not set up a Cs contour control mode switching signal.

NOTE
1. An attempt to turn ON the Cs contour control mode switching signal for a hypothetical Cs axis results in alarm SP0752.
2. If a program issues a move command for a hypothetical Cs axis for which composite control is OFF or for a hypothetical Cs axis and a Cs axis under composite control, alarm PS0197 will occur.

Example of control in which a hypothetical Cs axis and composite control are combined

1) Control in which a Cs contour control axis (a single serial spindle) is used in two paths

[Machining pattern]
Usually, machining is performed in path 1, using X1-Z1-C1.
By performing composite control on C1-C2, you can operate C1 (Cs axis) with a C2 command in path 2, thereby enabling machining using X2-Z2-C1.

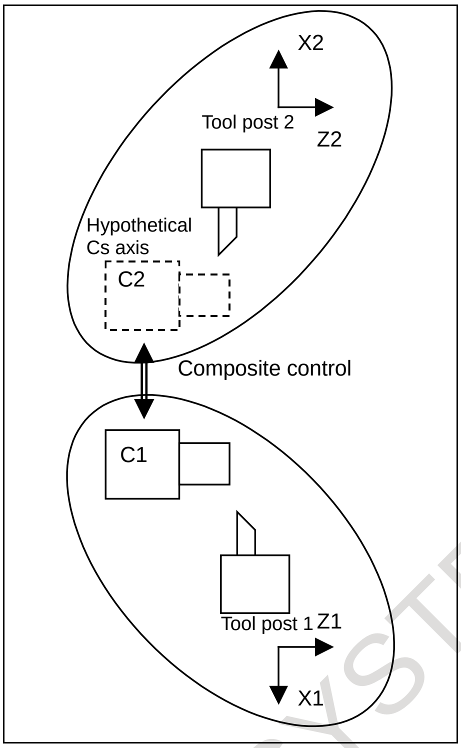


Fig. 8.6.3 (a)

Axis configuration

Path 1	Path 2
X1 (servo axis)	X2 (servo axis)
Z1 (servo axis)	Z2 (servo axis)
C1 (Cs-axis)	C2 (hypothetical Cs axis)

Parameter setting

		Parameter No. 1023 (servo axis number)	Bit 7 of parameter No. 1014 (Hypothetical Cs axis setting)
1st axis	Path 1, axis 1, X1	1	0
2nd axis	Path 1, axis 2, Z1	2	0
3rd axis	Path 1, axis 3, C1	-1	0
4th axis	Path 2, axis 1, X2	3	0
5th axis	Path 2, axis 2, Z2	4	0
6th axis	Path 2, axis 3, C2	-2	1

		Parameter No. 3717 (spindle motor number)
1st spindle	Path 1, spindle 1	1
2nd spindle	Path 2, spindle 1	0

2) Control in which Cs contour control axes (two serial spindle) are used in two paths

[Machining pattern]
Usually, machining is performed in path 1, using X1-Z1-C1, and in path 2, using X2-Z2-C2.
By performing composite control on B1-C2, you can operate C2 (Cs axis) with a B1 command in path 1, thereby enabling machining using X1-Z1-C2.
By performing composite control on C1-B2, you can operate C1 (Cs axis) with a B2 command in path 2, thereby enabling machining using X2-Z2-C1.

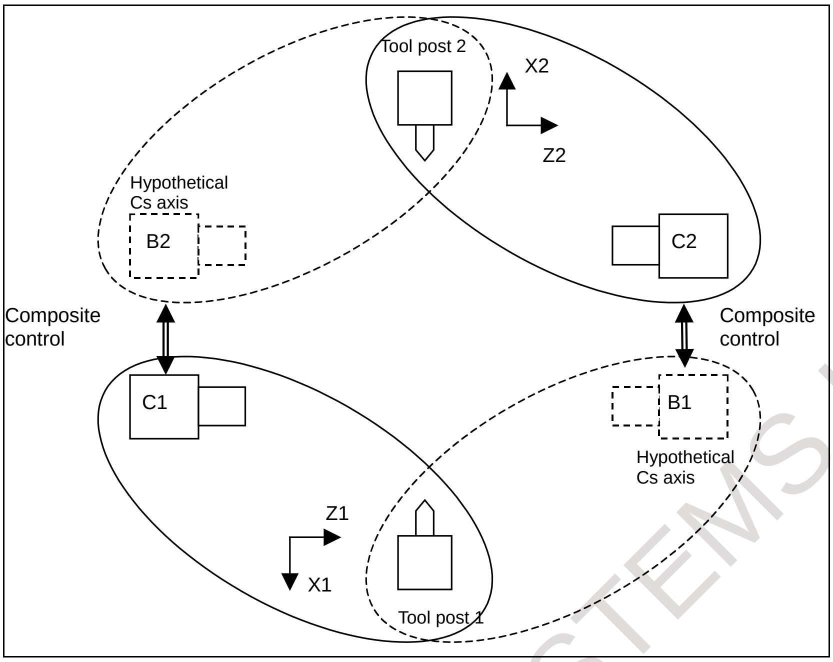


Fig. 8.6.3 (b)

Axis configuration

Path 1	Path 2
X1 (servo axis)	X2 (servo axis)
Z1 (servo axis)	Z2 (servo axis)
C1 (Cs-axis)	C2 (Cs-axis)
B1 (hypothetical Cs axis)	B2 (hypothetical Cs axis)

Configuration in which by using a hypothetical Cs axis, the addresses of commands for the Cs axis in the remote path are set separately from those of commands for the Cs axis in the local path.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1014	CDMx							

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#7 CDMx The Cs contour control axis is:
0: Not a virtual Cs axis
1: Virtual Cs axis

1023	Number of the servo axis for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input

[Data type] Byte axis
[Valid data range] 0 to 80

This parameter associates each control axis with a specific servo axis. Specify values 1+8n, 2+8n, 3+8n, 4+8n, 5+8n, and 6+8n (n = 0, 1, 2, …, 9) like 1, 2, 3, 4, 5, …, 77, and 78.
The control axis number is the order number that is used for setting the axis-type parameters or axis-type machine signals
With an axis for which Cs contour control/spindle positioning is to be performed, set -(spindle number) as the servo axis number.
Example)
When exercising Cs contour control on the fourth controlled axis by using the first spindle, set -1.

3717	Spindle amplifier number to each spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled axes

Set a spindle amplifier number to be assigned to each spindle.
0: No spindle amplifier is connected.
1: Spindle motor connected to amplifier number 1 is used.
2: Spindle motor connected to amplifier number 2 is used.
to
n: Spindle motor connected to amplifier number n is used.

Alarm and message

Number	Message	Description
PS0197	C-AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs-axis when the Cs contour control switching signal was off.
SP0752	SPINDLE MODE CHANGE ERROR	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contour control, spindle positioning, rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.

Caution

> ⚠ **CAUTION**
> 1 Set the necessary parameters such as feedrate for the hypothetical Cs axis as well. You need not set serial spindle parameters (such as Nos. 4000 to 4799).
> 2 For Cs contour control, see the explanation of each function.

Note

NOTE

This function requires the following functions.
- Serial spindle
- Cs contour control (bit 2 (SCS) of parameter No.8133 is set to 1)
- Synchronous/composite control

In the number of controlled axes specification and the number of controlled spindles specification, you must include the hypothetical Cs axis. (The maximum number of controllable axes and the maximum number of controllable spindles in the system or in each path must not be exceeded.)

8.7 SUPERIMPOSED CONTROL

Overview

The superimposed control function adds the amount of movement of an axis (superimposed control master axis) in one path to an axis (superimposed control slave axis) on the other path for which ordinary move commands are being executed. This function is similar to synchronous control but differs from it in that move commands can be issued not only for the master axis but also for the slave axis. The slave axis moves by the sum of the amount of movement specified by its own move commands and the amount of movement specified by move commands for the master axis. Appropriate setting of bit 3 (OMRx) of parameter No. 8162 can reverse the direction in which the master and slave axes move.

- Independent control of each path

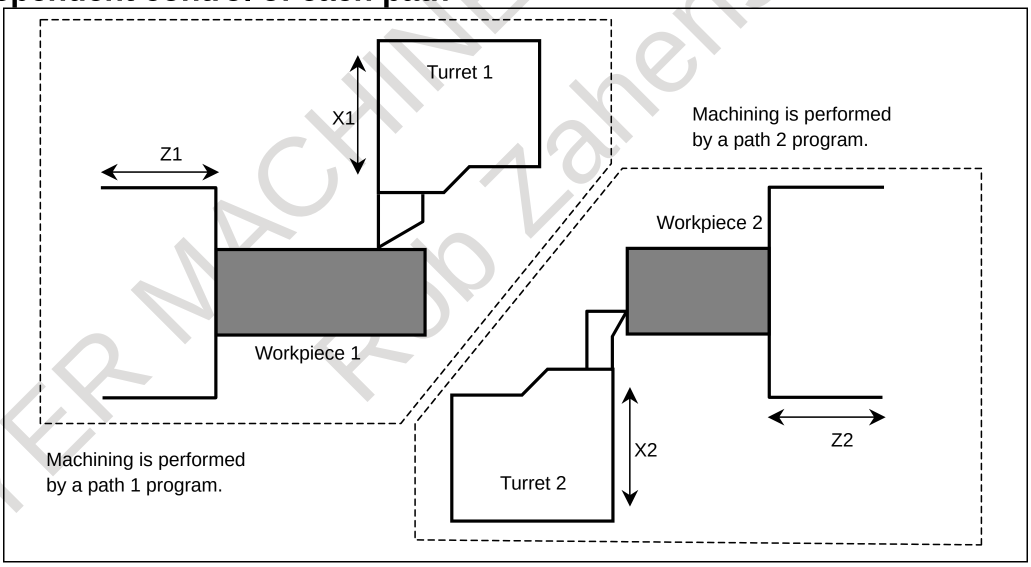


Fig. 8.7 (a)

This function enables superimposed control between paths and in a single path, as described below.

- Superimposed control

The move command for any axis is superimposed on an axis of another path.

(Example)

Superimpose the movement along the Z1 (master) axis on the Z2 (slave) axis.

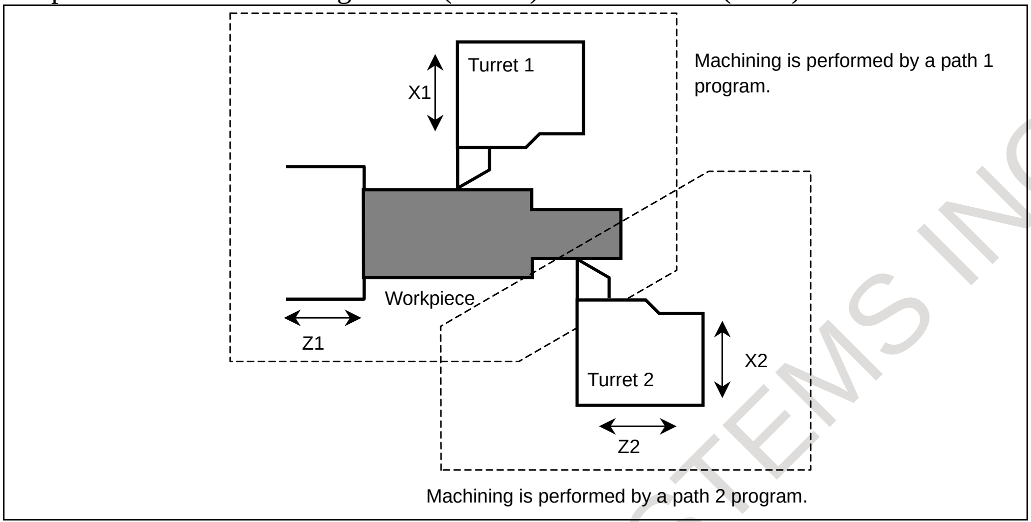


Fig. 8.7 (b)

Explanation

- Setting

Specify which axis is to be the master axis to be under superimposed control, with parameter No. 8186, using the path number and the axis number.

Example)

For an axis configuration in which all paths consist of X, Z, and Y axes
To superimpose the movement of the Z1-axis (master) to that of the Z2-axis (slave) :
Parameter No. 8186z of path 2 = 102
To superimpose the movement of the X3-axis (master) to that of the Z4-axis (slave) :
Parameter No. 8186z of path 4 = 301
To superimpose the movement of the X1-axis (master) to that of the Y1-axis (slave) :
Parameter No. 8186y of path 1 = 101

- Programming

Both before and after the M codes for a start and a cancellation of superimposed control, specify a waiting M code.

Master axis path	Slave axis path	
:	:	
M100P12;	M100P12;	Waiting
	M220;	Start of superimposed control
M101P12;	M101P12;	Waiting
:	:	
Independent operation	Independent operation	
:	:	
M100P12;	M100P12;	Waiting
	M221;	Cancellation of superimposed control
M101P12;	M101P12;	Waiting
:	:	

- Signal operation

When superimposed control begins or ends (when an M code is issued), the superimposed control axis selection signals OVLS1 to OVLS8 <Gn190> for the target slave axis (from the PMC to the CNC) are changed from "0" to "1" (to begin superimposed control) or from "1" to "0" (to terminate superimposed control).

- Examples of applications

Suppose that a workpiece on the spindle (Z1-axis) that moves along the axis is to be cut with a tool in path 1 and a tool in path 2 simultaneously. This example superimposes the amount of movement of the Z1-axis (master) on that of the Z2-axis (slave).

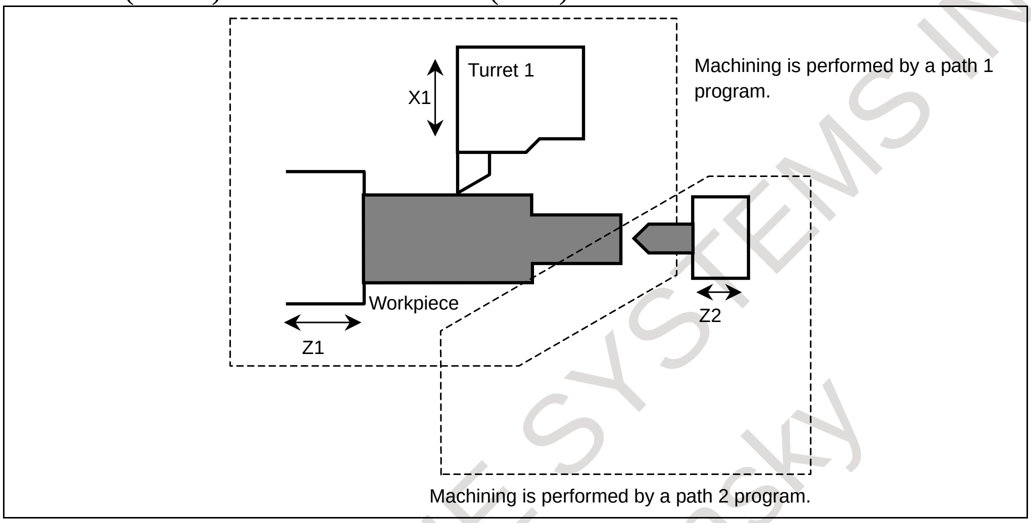


Fig. 8.7 (c)

- Feedrate

Because the amount of movement of the master axis is added to that of the slave axis, the resulting speed of the slave axis may become much larger than a normal speed.

- Differences between superimposed control and ordinary synchronous control

- Neither out-of-synchronization compensation nor detection is performed between the master and slave axes during superimposed control.
- A parking signal is ineffective for axes under superimposed control.
- A reference position return cannot be specified for the salve axis under superimposed control.

Signal

Superimposed control axis selection signals OVLS1 to OVLS8 <Gn190>

[Classification] Input signal

[Function] These signals perform superimposed control.

[Operation] When one of these signals becomes "1", the control unit:

- Begins superimposed control over the corresponding axis.
  The master axis is selected according to parameter No. 8186.

Synchronous/composite/superimposed control under way signals SYN1O to SYN8O <Fn118>

[Classification] Output signal

[Function] These signals indicate each axis is being subjected to synchronous, composite, or superimposed control.

[Output cond.] These signals become "1" under the following condition:

- The corresponding axis is under synchronous, composite, or superimposed control.

These signals become "0" under the following condition:

- The corresponding axis is not under synchronous, composite, or superimposed control.

> ⚠ **CAUTION**
> Whether each axis is under superimposed control does not always match whether the superimposed control axis selection signal has been issued or not. For example, if these signals are set to 1 during an alarm, they are ignored. If a servo alarm occurs during superimposed control, they are terminated automatically. Before attempting to perform superimposed control, always check the state of these signals.

Superimposed control master axis confirmation signals OVMO1 to OVMO8 <Fn344>

[Classification] Output signal

[Function] These signals notify whether the corresponding axes are superimposed control master axes.

[Output cond.] These signals become "1" under the following condition:

- The corresponding axes are superimposed control master axes.

These signals become "0" under the following condition:

- The corresponding axes are released from superimposed control.

Superimposed control slave axis confirmation signals OVSO1 to OVSO8 <Fn345>

[Classification] Output signal

[Function] These signals notify whether the corresponding axes are superimposed control slave axes.

[Output cond.] These signals become "1" under the following condition:

- The corresponding axes are superimposed slave axes.

These signals become "0" under the following condition:

- The corresponding axes are released from superimposed control.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn190	OVLS8	OVLS7	OVLS6	OVLS5	OVLS4	OVLS3	OVLS2	OVLS1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn118	SYN8O	SYN7O	SYN6O	SYN5O	SYN4O	SYN3O	SYN2O	SYN1O
Fn344	OVMO8	OVMO7	OVMO6	OVMO5	OVMO4	OVMO3	OVMO2	OVMO1
Fn345	OVSO8	OVSO7	OVSO6	OVSO5	OVSO4	OVSO3	OVSO2	OVSO1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8160	NRS		NCS	AXS				

[Input type] Parameter input

[Data type] Bit path

#4 AXS When the axis moving signals MV1 to MV8<Fn102> or the axis moving direction signals MVD1 to MVD8<Fn106> of the slave axis in superimposed control is output:

0: State output is performed according to the result of adding superimposed move pulses.

1: State output is performed according to the result of movement along each axis instead of superimposed move pulses.

#5 NCS If an overtravel alarm is issued for an axis under synchronous, composite, or superimposed control, synchronous, composite, or superimposed control is:
0: Released.
1: Not released.

> **NOTE**
> If this parameter is set to 1 for any one of the paths in a machine group, it is assumed to be set to 1 for all the paths.

#7 NRS When the system is reset, synchronous, composite, or superimposed control is:
0: Released.
1: Not released.

	#7	#6	#5	#4	#3	#2	#1	#0
8162					OMRx			

[Input type] Parameter input
[Data type] Bit axis

#3 OMRx Superimposed mirror-image control is:
0: Not applied. (The superimposed pulse is simply added.)
1: Applied. (The inverted superimposed pulse is added.)

	#7	#6	#5	#4	#3	#2	#1	#0
8164		SOKx	OPSx					

[Input type] Parameter input
[Data type] Bit axis

#5 OPSx When superimposed control is canceled, control in which an amount of movement along a master axis subject to superimposed control is added to the workpiece coordinate of a slave axis is:
0: Not applied.
1: Applied.

> **NOTE**
> When the workpiece coordinate system is enabled (bit 0 of parameter No.8136 is 0), workpiece coordinate system presetting (equivalent to G92.1IP0) is performed to set up a coordinate system.

#6 SOKx If a master axis subject to superimposed control is also subject to synchronous control:
0: An alarm is issued when superimposed control is started during synchronous control.
1: No alarm is issued when superimposed control is started during synchronous control.

	#7	#6	#5	#4	#3	#2	#1	#0
8168				MWR			MSO	MPA

[Input type] Parameter input

[Data type] Bit

#0 MPA If an alarm concerning synchronous control, composite control, or superimposed control is issued:

0: All paths of the machine group to which the alarm occurrence path belongs are placed in feed hold state.

1: Only the path including the axis placed under synchronous control, composite control, or superimposed control is placed in the feed hold state.

#1 MSO When one of the following events occurs in synchronous control, composite control or superimposed control:

- The emergency stop signal *ESP <Gn008.4> is turned off.
- The servo-off signals SVF1 to SVF8 <Gn126.0 to Gn126.7> are turned on.
- A servo alarm is issued.

0: The synchronous, composite control or superimposed control mode is canceled and follow-up operation is not performed.
For the operation to be performed when the servo-off signal is turned on, however, the setting of bit 7 (NSR) of parameter No. 8161 is used in synchronous control or the setting of bit 0 (NMR) of parameter No. 8161 is used in composite control.

1: The synchronous, composite control or superimposed control mode is not canceled. The following operation is performed to perform follow-up operation:
When the emergency stop signal *ESP is turned off, the relevant path is determined and operation is performed so that the emergency stop signal *ESP is virtually turned off for the determined path.
When the servo-off signals SVF1 to SVF8 are turned on, the relevant axis is determined and operation is performed so that the servo-off signals SVF1 to SVF8 are virtually turned on for the determined axis.
When a servo alarm is issued, the relevant axis is determined and the alarm SV0003, "SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL MODE CAN'T BE CONTINUED" is issued for the determined axis to stop moving the tool along the axis. When bit 2 (SVF) of parameter No. 8168 is set to 1, this servo-off specification follows the SVF setting.

> **NOTE**
> 1 This setting is valid also during operation. For all axes placed under synchronous, composite control or superimposed control, the emergency stop signal is turned off, the servo-off signal is turned on, or a servo alarm is issued.
> 2 If the servo-off signal is turned on, alarm DS1933 "NEED REF RETURN (SYNC:MIX:OVL)" will be generated in superimposed control.
> After canceling alarm, perform a manual reference position return.

#4 MWR When the synchronous control, composite control, or superimposed control is started or ended and the automatic setting of a workpiece coordinate system is executed, the tool offset number is:

0: Canceled.

1: Not canceled.

> **NOTE**
> This parameter is valid when the workpiece coordinate system is enabled (bit 0 of parameter No.8136 is 0).

8186	Master axis under superimposed control

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 101, 102, 103,…, path number *100 + intra-path relative axis number (101, 102, 103,…, 201, 202, 203,…, 1001, 1002, 1003,….)

This parameter sets the path number and intra-path relative axis number of a superimposed master axis for each axis when superimposed control is exercised. When zero is specified, the axis does not become a slave axis under superimposed control and the move pulse of another axis is not superimposed.
An identical number can be specified in two or more parameters to exercise superimposed control simultaneously. This means that superimposed control with one master axis and multiple slave axes is possible.
A slave axis may function as the master axis of another axis to allow three-generation superimposed control: parent (master axis) - child (slave axis/master axis) - grandchild (slave axis).

In this case, a movement along the child is made by its travel distance plus the travel distance of the parent, and a movement along the grandchild is made by its travel distance plus the travel distance of the child plus the travel distance of the parent.

Example of the relationship of parent (X1 of path 1) - child (X2 of path 2) - grandchild (X3 of path 3):
The travel distance of X1 is superimposed on X2, and the travel distances of X1 and X2 are further superimposed on X3.
Parameter No. 8186 (X axis) of path 2 = 101
Parameter No. 8186 (X axis) of path 3 = 201

	#7	#6	#5	#4	#3	#2	#1	#0
11284								SSH

[Input type] Parameter input
[Data type] Bit

 # 0 SSH During superimposed control, manual handle interruption to the slave axis is:
0: Disabled.
1: Enabled.

Alarm and message

If any of the following alarms occurs, superimposed control will be released for all axes.

- P/S alarm

Number	Message	Description
PS0360	PARAMETER OF THE INDEX OF THE SUPERPOS CONTROL AXIS SET ERROR.	An illegal superimposed control axis number (parameter No. 8186) is specified.
PS0361	BECAUSE THE AXIS IS MOVING, THE SUPERPOS CONTROL IS CAN'T BE USED.	While the axis being subject to superimposed control was moving, an attempt was made to start or cancel the superimposed control by a superimposed control axis selection signal.

Number	Message	Description
PS0362	SUPERPOSITION CONTROL AXIS COMPOSITION ERROR.	This error occurred when: 1) An attempt was made to perform superimposed control for the axis in synchronization, composition, or superimposed. 2) An attempt was made to synchronize a further great-grandchild for a parent-child-grandchild relation.
PS0363	THE G28 WAS INSTRUCTED IN TO THE SUPERPOS CONTROL SLAVE AXIS.	This error occurred when G28 was specified to the superimposed control slave axis during superimposed control.
PS0364	THE G53 WAS INSTRUCTED IN TO THE SUPERPOS CONTROL SLAVE AXIS.	In superimposed control, this error occurs if G53 is specified to the slave axis during movement along the master axis.

- D/S alarm

Number	Message	Description
DS1933	NEED REF RETURN(SYNC:MIX:OVL)	The relation between a machine coordinate of an axis in synchronization, composition, or superimposed control, and the absolute, or relative coordinate was displaced. Perform the manual return to the reference position.

Caution

⚠ CAUTION

1 At the start or end of superimposed control, the tool must be stopped on the axis subject to that control.
2 Before or after an M code for starting or canceling superimposed control during automatic operation, be sure to specify a waiting M code (M code without buffering). When exercising superimposed control in a path, be sure to prohibit look-ahead operation by specifying an M code without buffering before or after an M code for starting or canceling superimposed control.
3 The axes under superimposed control must match in least command increment, detection unit, and diameter/radius specification. Otherwise, the amounts of travel will not be equal.
4 In superimposed control, do not change the parameters related to it (including axis control, increment system, feedrate, and acceleration/deceleration control).
5 Before starting superimposed control, make sure that the axes subject to it has undergone a reference position return after the power was turned on or that a reference position has been established with an absolute pulse coder.
6 If an emergency stop or servo off is performed or if a servo alarm occurs, the coordinates will change due to superimposed. If this occurs, alarm DS1933, "NEED REF RETURN (SYNC:MIX:OVL)" will be generated. After canceling the emergency stop, canceling servo off, or restoring from the servo alarm, perform a reference position return and coordinate system setting first, then start superimposed control.
7 In superimposed control, pitch error compensation, backlash compensation, and stored stroke check are performed on the position resulting from adding superimposing pulses.

⚠ **CAUTION**

8 The following servo software functions cannot be used with superimposed control:
- Feed forward function(*)
- Advance preview feed forward function(*)
- Fine acceleration/deceleration
- Abnormal load detection and switching function for each of cutting and rapid traverse
- Fine acceleration/deceleration and switching function for each of cutting and rapid traverse
- Cutting/rapid velocity gain switching function
- Electric current 1/2 and PI switching function for each of cutting and rapid traverse
- Torque command filter switching function for each of cutting and rapid traverse

9 The following functions cannot be used in superimposed control:
- Electronic gear box
- AI contour control I/II(*)
- Tool retract and recover

10 During superimposed control, reference position establishment based on "linear scale with absolute address reference mark" or "linear scale with absolute address zero point" is impossible.

(*) Please refer the section of "SUPERIMPOSED CONTROL AVAILABLE IN THE AI CONTOUR CONTROL MODE".

Note

NOTE

1 You can place more than one axis under superimposed control at the same time. You cannot, however, place an axis subject to synchronous or composite control under superimposed control.

2 Superimposed control cannot be performed on a linear axis and a rotation axis.

3 The superimposed control and axis synchronous control option can be specified at the same time. In this case, the master axis for superimposed control can be specified as the master axis for axis synchronous control.

Limitation

- Limitations on superimposed control

Function	In superimposed control
Acceleration/deceleration control	The move pulses of the superimposed control master axis after acceleration/deceleration are added to those of the slave axis after acceleration/deceleration.
Reference position return	Not allowed for a superimposed slave axis.
2nd, 3rd, 4th reference position return	Not allowed for a superimposed slave axis.
Reference position return check	Not allowed for a superimposed slave axis.
Machine coordinate system selection	Not allowed for a superimposed slave axis. (3)
PMC axis control	Allowed.
Polar coordinate interpolation, Cylindrical interpolation	Allowed.
Handle interruption	Not allowed for a superimposed control axis. (4)
Mirror image	Only master signals are effective to superimposing pulses. (1)
Machine lock	Only master signals are effective to superimposing pulses. (1)

Function	In superimposed control
Inter lock	Only master signals are effective to superimposing pulses. (1)
Override	Only master signals are effective to superimposing pulses. (1)
External deceleration	Only master signals are effective to superimposing pulses. (1)
Skip function	Allowed.
Automatic tool offset	Not allowed for a superimposed slave axis.
Follow-up	Not allowed in superimposed control.
Program restart	Not allowed for program containing superimposed control.
Cs contour control	Superimposed control is possible. (2)
Spindle positioning	Superimposed control is impossible.
EGB function	Not allowed in superimposed control.
Servo off	Superimposed control is impossible.

(1) For the move commands inherent to the master and slave, these functions are processed as usual; for superimposed pulses from the master axis, these functions are not applied to the slave.
(2) Limited to combinations of Cs axes. Switch to Cs mode and perform a reference position return first, then start superimposed control.
(3) If there are no superimposed pulses from the master axis, the machine coordinate system selection command can be specified on the path containing the slave axis.
Example)
If X1 is a master axis, and X2 is a slave axis

Path 1	Path 2	
M120P12;	M120P12;	Waiting
G01Z100.0;	G53X50.0;	X1 (master axis)-X2 (slave axis) superimposed control
M121P12;	M121P12;	Waiting

In this case, alarm PS0364 is not issued because there is no command for X1, which is the master axis of X2 on path 2.
(4) If bit 0 (SSH) of parameter No. 11284 is set to 1, handle interrupt can be performed on the slave axis.

- Reading of coordinates in superimposed control

In superimposed control, the reading of custom macro system variable positional information or of current coordinates from the PMC window is as follows:

Positional information type	In superimposed control
Absolute coordinate	Reading possible. (*1)
Machine coordinate	Reading possible.
End point of each block	Reading possible. (*1)
Skip signal position	Reading possible. (*1)

(*1) Superimposing pulses are not added.

- Canceling superimposed control

Superimposed control is canceled in the event of the following, as well as when the synchronous/composite/superimposed control axis selection signal becomes "0".
(1) Emergency stop
(2) Reset
(3) Servo alarm
(4) Servo off
(5) Overtravel *1
(6) Alarm related to superimposed control
(7) Alarm PW0000
If one of the above events occurs in either path, all paths will be released from superimposed control. If one of the above events occurs in either path in superimposed control, the other path will automatically be placed in feedhold state (during automatic operation) or in interlock state (during manual operation).

*1: By setting bit 5 (NCS) of parameter No. 8160 to 1, you can prohibit the superimposed control axis from release from the superimposed control state even if the axis enters the overtravel state.

- Axis state output signals in superimposed control

State output signal	In superimposed control
Axis moving signals MVn <Fn102>	- On the master axis, as usual. - The slave axis reflects the travel state with the command on the slave, regardless of superimposed control pulses. (*1)(*2).
Axis moving direction signals MVDn <Fn106>	- On the master axis, the moving direction of the master axis. - For the slave axis, the moving direction after superimposed control pulses are added. (*2).
In-position signals INPn <Fn104>	- On the master axis, as usual. - The slave axis signal is always "1".
Reference position establishment signal ZRFn <Fn120>	- The signal on the axis for which the reference position has been established turns to "1".
Reference position return completion signal ZPn <Fn094>	- The signal value for each axis depends on the state of the axis.

(*1) In a positional deviation check, regardless of the states of these signals, parameter No. 1828 is used if move command pulses are sent to the motor (regardless of whether the axis is the master or slave) as the limit, and parameter No. 1829 if none are sent.

(*2) Using bit 4 (AXS) of parameter No. 8160, you can switch the slave axis state output between results with superimposed control pulses added and results of individual axial movements.

Examples of Use

- Examples of independent control and of superimposed control on the Z1 and Z2 axes

(1) Machine configuration

(a) Independent control

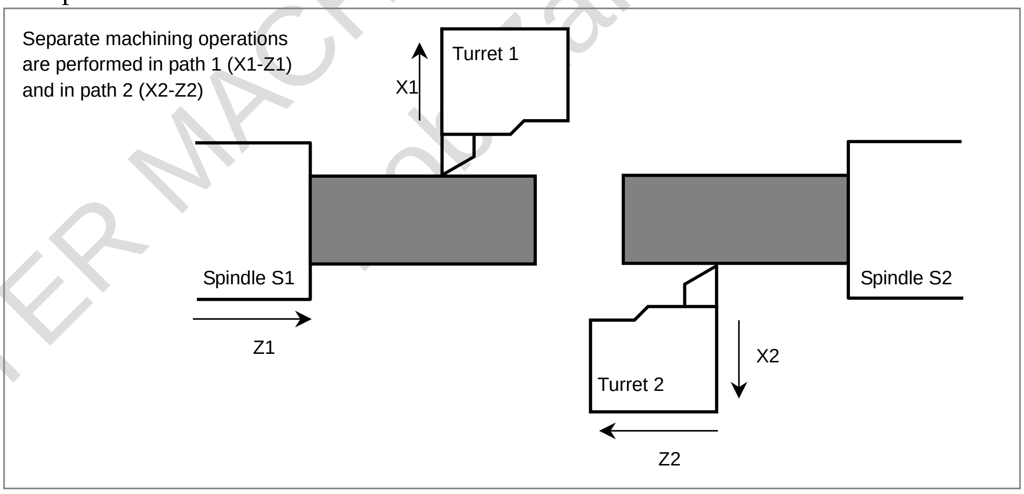


Fig. 8.7 (d)

(b) Z1-Z2 axis superimposed control

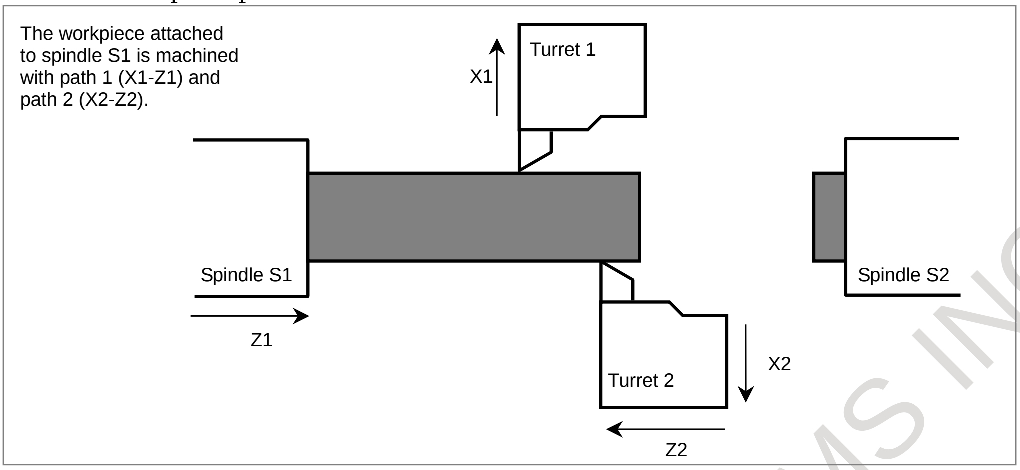


Fig. 8.7 (e)

(2) Parameter setting

- To superimpose the move command for the Z1 (master) axis on that of the Z2 (slave) axis, set parameter No. 8186z of path 2 to "102".
- Since the plus directions of the Z1 (master) and Z2 (slave) axes are opposite from each other, perform superimposed control with a mirror image applied. To do this, set bit 3 (OMRz) of parameter No. 8162 of path 2 to 1.

(3) Signal operation

- To start superimposed movement along the Z1 (master) axis and the Z2 (slave) axis, set signal OVLS2 <G1190.1> to "1".
- To cancel superimpose, set signal OVLS2 <G1190.1> to "0".
- When an emergency stop, NC reset, or alarm occurs, set signal OVLS2 <G1190.1> to "0".
- Set signals other than signal OVLS2 <G1190.1> to "0".

(4) Program example

Path 1		Path 2		
N1000	. . .	N2000	. . .	Independent machining of each path
N1010	M200 P12 ;	N2010	M200 P12 ;	Waiting
		N2020	M55 ;	Start of superimposed control from the Z1 axis to the Z2 axis
N1030	M201 P12 ;	N2030	M201 P12 ;	Waiting
		N2040	T0414 ;	Selection of a superimposed control tool and setting of an offset
N1050	S1000 M3 ;			
N1060	G0 X20.0 Z15.0 ;	N2060	G0 X18.0 Z120.0 ;	Machining by turret 1 and turret2
N1070	G1 F0.5 W-8.0 ;	N2070	G1 F0.1 W5.0 ;	
N1080	. . .	N2080	. . .	
N1010	M200 P12 ;	N2010	M200 P12 ;	Waiting
		N2100	M56 ;	End of superimposed control
N1010	M201 P12 ;	N2010	M201 P12 ;	Waiting
N1120	. . .	N2120	. . .	Independent machining of each path

where
M55 is an M code to start superimposed control, and
M56 is an M code to cancel superimposed control.

> ⚠ **CAUTION**
> If performing constant surface speed control, you must determine the spindle command in which path is currently effective to spindle S1.

> **NOTE**
> Input the speed of spindle S1 (feedback pulses from the position coder) to both paths 1 and 2.

Troubleshooting

1. Unable to start superimposed control. (No alarm occurs.)
   (1) The correct superimposed control have not been set up.
   → To perform superimposed control, you must set up the correct
   (2) The superimposed control axis selection signals OVLS1 to OVLS8 <Gn190> is not set to "1".
   → Superimposed control starts when the superimposed control axis selection signals OVLS1 to OVLS8 <Gn190> is changed from "0" to "1". After superimposed control ends due to a reset or alarm, merely canceling the reset or alarm cause will not cause superimposed control to restart. After canceling the reset or alarm cause, you must make the signal rise again.
   (3) The axis number of the axis subject to superimposed control is not set in the appropriate parameter.
   → To perform superimposed control, set the axis number of the master axis in parameter No. 8186.

(4) Superimposed control cannot be started if the NC unit is one of the following states:
- Emergency stop
- Reset
- Servo alarm
- Alarm PW0000
- Alarm related to superimposed control

Superimposed control cannot be started, either, if the axis on which superimposed control is to start is either of the following states:
- Servo off
- Overtravel

2. An attempt to raise a superimposed start signal results in PS alarm.
   (1) An attempt is made to use an axis already under superimposed control for another superimposed signal operation. (Alarm PS0362)
       → You cannot use a single axis for a combination of multiple superimposed control operations.
   (2) The axis number set in a parameter is greater than the number of controllable axes. (Alarm PS0360)
       → The number of the superimposed master axis and that of the interchanging axis subject to superimposed control must not exceed the number of axes controllable in the remote path (or the local path if you are to perform superimposed control in that path).
   (3) The tool is moving along the axis that you want to control. (Alarm PS0361)
       → At the start of superimposed control, the tool must be stopped on the axis subject to that control. That the tool is stopped on an axis means that the speed specified after acceleration/deceleration is zero.
   (4) An axis on which you want to perform superimposed control belongs to another machine group. (Alarm PS0360)
       → The axes on which you want to perform superimposed control must be of the same machine group.

3. An attempt to cancel superimposed control results in alarm PS0361.
   (1) The tool is moving along the axis on which the control you want to cancel.
       → To cancel superimposed control, the tool must be stopped on the axis on which to cancel it. That the tool is stopped on an axis means that the speed specified after acceleration/deceleration is zero. Before canceling, make sure that the axis moving signals MV1 to MV8 <Fn102> has changed to "0".

4. In superimposed control, the amount of travel is not correct.
   (1) The master and slave axes differ in diameter/radius specification or in inch/millimeter setting.
       → In superimposed control, diameter/radius conversion or inch/millimeter conversion is not performed between master and slave. The master and slave must match in least command increment.

5. In superimposed control, alarm PS0363 occurs.
   (1) In superimposed control, a reference position return is performed on the slave axis.
       → No reference position return can be performed on the slave axis under superimposed control.

6. Superimposed control is canceled although the superimposed control axis selection signals OVLS1 to OVLS8 <Gn190> is not changed from "1" to "0".
   → In superimposed control, superimposed control will be automatically canceled if at least either of the paths enters one of the following states:
   - Emergency stop
   - Reset
   - Servo alarm
   - Alarm PW0000
   - Alarm related to superimposed control

   In addition, if at least one of the axes on which superimposed control is in progress enters either of the following states:
   - Servo off
   - Overtravel

7. Alarm DS1933 occurs.
   → In superimposed control, alarm DS1933 occurs if at least one path enters one of the following states:
   - Emergency stop
   - Servo off
   - Servo alarm

   After canceling the emergency stop, after canceling servo off, or after restoring from the servo alarm, first perform a reference position return or coordinate system setting and then start superimposed control.

8.8 SUPERIMPOSED CONTROL(WITH SPEED CONTROL)

For a slave axis under superimposed control, a travel distance specified by the program for the master axis path is added to a travel distance specified by the program for the slave axis path. So, the actual speed on a slave axis is excessively higher than ordinary speed ("ordinary speed" means a speed such as a parameter-set rapid traverse rate).
To prevent this, feedrates and a linear acceleration/deceleration time constant in rapid traverse can be set for use only during superimposed control with this function.

The parameters for feedrates and a time constant usable only during superimposed control are indicated below.

-	Rapid traverse rate	Parameter No. 8190
-	F0 velocity of rapid traverse override	Parameter No. 8191
-	Linear acceleration/deceleration time constant in rapid traverse	Parameter No. 8192
-	Maximum cutting feedrate	Parameter No. 8194
(-	Manual rapid traverse rate	Value of parameter No. 8190 or 1424, whichever smaller)

During superimposed control, each of the master and slave axes uses these separately set parameters.
Set proper values, considering a feedrate after addition of a travel distance.

Parameter

8190	Rapid traverse rate of an axis under superimposed control

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to standard parameter setting table (C) (See “PREFACE” in this manual.)
(When the increment system is IS-B, 0 to +999000.0)

Set a rapid traverse rate for each of the axes when the rapid traverse override of the axes (master and slave axes) under superimposed control is 100%. The manual rapid traverse rate set in this parameter or the manual rapid traverse rate set in parameter No. 1424, whichever smaller, is used.
If this parameter is set to 0, the normal rapid traverse rate (parameter No. 1420) is used.

8191	F0 velocity of rapid traverse override of an axis under superimposed control

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to standard parameter setting table (C) (See “PREFACE” in this manual.)
(When the increment system is IS-B, 0 to +999000.0)
Set the F0 velocity of rapid traverse override of an axis under superimposed control (each of the master and slave axes).
If this parameter is set to 0, the F0 velocity of rapid traverse override in normal operation (parameter No. 1421) is used.

8192	Linear acceleration/deceleration time constant in rapid traverse of an axis under superimposed control

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
This parameter specifies the linear acceleration/deceleration time constant in rapid traverse for each of the axes (master and slave axes) under superimposed control.

8194	Maximum cutting feedrate in superimposed control

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to standard parameter setting table (C) (See “PREFACE” in this manual.)
(When the increment system is IS-B, 0 to +999000.0)
Set the maximum cutting feedrate that can be applied under superimposed control.
If this parameter is set to 0, the maximum cutting feedrate in normal operation (parameter No. 1430) is used.

8.9 SYNCHRONOUS, COMPOSITE, AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND

Overview

Synchronous control, composite control, and superimposed control can be started or canceled using a program command instead of a DI signal.
Synchronous control, composite control, and superimposed control based on a DI signal is also possible.
For the basic operations of synchronous control, composite control, and superimposed control, see Section "SYNCHRONOUS / COMPOSITE CONTROL", and "SUPERIMPOSED CONTROL".

Format

G51.4 P_ Q_ (L_) ; Start synchronous control (L_ can be omitted)
G50.4 Q_ ; Cancel synchronous control

P: Number to identify synchronous master axis
Q: Number to identify synchronous slave axis
L: Parking state command
1: Master parking (slave parking cancel)
2: Slave parking (master parking cancel)
0: No parking (parking cancel)
(When L is omitted, the specification of L0 is assumed.)

G51.5 P_ Q_ ; Start composite control
G50.5 P_ Q_ ; Cancel composite control

P: Number to identify composite axis 1
Q: Number to identify composite axis 2

G51.6 P_ Q_ ; Start superimposed control
G50.6 Q_ ; Cancel superimposed control

P: Number to identify superimposed master axis
Q: Number to identify superimposed slave axis

Identification numbers are unique values that set into parameter No. 12600 in order to identify each axes.

NOTE
1 These commands are one-shot G code.
2 These commands must be specified in a single block.
3 When P,Q or L command is out of range, the alarm PS5339"ILLEGAL FORMAT COMMAND IS EXECUTED IN SYNC/MIX/OVL CONTROL." occurs.

Explanation

- Synchronous control

Synchronous control is exercised by the G51.4/G50.4 commands instead of the synchronous control axis selection signals (SYNC1 to SYNC8<Gn138.0-7>, n=0 to 9).

- Composite control

Composite control is exercised by the G51.5/G50.5 commands instead of the composite control axis change selection signals (MIX1 to MIX8<Gn128.0-7>, n=0 to 9).

- Superimposed control

Superimposed control is exercised by the G51.6/G50.6 commands instead of the superimposed control axis selection signals (OVLS1 to OVLS8<Gn190.0-7>, n=0 to 9).

Parameter

12600	Identification Number for synchronous, composite, and superimposed control by program command

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0,1 to 32767

Set identification numbers that can be specified with P,Q addresses.
The axis whose identification number is 0 cannot become under synchronous, composite, and superimposed control by program command.
The same identification number cannot be set to two or more axes through all paths.
When the same identification number is set, alarm PS5339 occurs at G50.4/G50.5/G50.6/G51.4/G51.5/G51.6 commands.

Alarm and message

Number	Message	Description
PS5339	ILLEGAL FORMAT COMMAND IS EXECUTED IN SYNC/MIX/OVL CONTROL.	1. The value of P, Q, or L specified by G51.4/G50.4/G51.5/G50.5/G51.6/G50.6 is invalid. 2. A duplicate value is specified by parameter No. 12600.

Note

NOTE

1 If a G code (G50.4/G50.5/G50.6) for canceling synchronous, composite, or superimposed control based on a program command is specified using a DI signal for an axis under synchronous, composite, or superimposed control, synchronous, composite, or superimposed control is canceled.

2 If the synchronous control axis selection signal/composite control axis change selection signal/superimposed control axis selection signal is set to 1 from 0 and to 0 from 1 for an axis under synchronous, composite, or superimposed control based on a program command, synchronous, composite, or superimposed control is canceled.
For an axis under synchronous, composite, or superimposed control based on a program command, if the synchronous control axis selection signal/composite control axis change selection signal/superimposed control axis selection signal is set to 1 from 0, the state of synchronous, composite, or superimposed control isn't influenced.

8.10 SUPERIMPOSED CONTROL AVAILABLE IN THE AI CONTOUR CONTROL MODE

Overview

This function enables superimposed control in the AI contour control mode.
Moreover, advance preview feed forward function becomes effective by the AI contour control when the parameters about advance preview feed forward function are set.

Explanation

To use superimposed control and AI contour control simultaneously during automatic operation, it is necessary to set the AI contour control permission signal OVLN to 1 and place the CNC in the advanced superimposition mode.

NOTE

1. To use superimposed control in the AI contour control mode, set the AI contour control permission signal for the master and slave axis paths to 1. If superimposed control is started when the advanced superimposition mode is off, AI contour control is turned off. Moreover, lowering of the precision or the shock might occur when the advanced superimposition mode for the one of paths is off.
2. Setting the AI contour control permission signal to 1 immediately after power-on can place the master and slave axis paths in the advanced superimposition mode without specifying a superimposed control mode on command in the program.

See the following example for the program sequence for specifying M codes for the advanced superimposition mode and superimposed control.

Program example)

Path of master axis	Path of slave axis
M100P12; ----- Waiting M210; ----- Advanced superimposition mode on. (Note 1) M101P12; ----- Waiting G05.1Q1; ----- AI contour control mode on. (Note 2) ~	M100P12; ----- Waiting M212; ----- Advanced superimposition mode on. M60; ----- Starts superimposed control. M101P12; ----- Waiting G05.1Q1; ----- AI contour control mode on. ~
: ------ Master axis move commands	~
~ M102P12; ----- Waiting M211; ----- Advanced superimposition mode off. (Note 1) M103P12; ----- Waiting	~ M102P12; ----- Waiting M61; ----- Cancels superimposed control. M213; ----- Advanced superimposition mode off. M103P12; ----- Waiting

NOTE

1. Turn the advanced superimposition mode on or off after canceling superimposed control for the slave path.
2. The AI contour control mode can be turned on by setting bit 0 (SHP) of parameter No. 1604 to 1 instead of using G05.1Q1;.

(1) Specify the waiting M code before and after the M codes for starting advanced superimposition and superimposed control.
  (a) Starts advanced superimposed control.
  Set the AI contour control permission signal OVLN to 1 and confirm that the advanced superimposition mode signal OVLNS is changed to 1.
  (b) Starts superimposed control.
  Set the superimposed control axis selection signal OVLS1 to OVLS8 for the slave axis of superimposed control to 1 (start of superimposed control) and confirm that the superimposed control under way signal SYN1O to SYN8O is changed to 1.

(2) Specify the waiting M code before and after the M codes for canceling advanced superimposition and superimposed control.
  (a) Cancels superimposed control.
  Set the superimposed control axis selection signal OVLS1 to OVLS8 for the slave axis of superimposed control to 0 (cancel of superimposed control) and confirm that the superimposed control under way signal SYN1O to SYN8O is changed to 0.
  (b) Cancels advanced superimposed control.
  Set the AI contour control permission signal OVLN to 0 and confirm that the advanced superimposition mode signal OVLNS is changed to 0.

Functions for which the advanced superimpositon mode cannot be specified

For any path for which the advanced superimposition mode is on, the following functions cannot be specified.
If any of the following functions is specified in the advanced superimposition mode, alarm PS0502 is issued.

- Reference position return in Cs contour control (G00,G28)
- Skip function (G31)
- Automatic tool length measurement / Automatic tool offset
- Low-speed type automatic reference position return (G28)
- High-speed program check

These functions can be specified after the advanced superimposition mode and superimposed control are turned off.
See the following example for the program sequence for specifying M codes for the advanced superimposition mode and superimposed control when a function that cannot be specified in the advanced superimposition mode is used.

Program example) (specifying G31 for the slave axis)

Path of master axis	Path of slave axis
M100P12; ----- Waiting M210; ----- Advanced superimposition mode on. M101P12; ----- Waiting G05.1Q1; ----- AI contour control mode on. ~	M100P12; ----- Waiting M212; ----- Advanced superimposition mode on. M60; ----- Starts superimposed control. M101P12; ----- Waiting G05.1Q1; ----- AI contour control mode on. ~
: ------ Master axis move commands	~
~ M102P12; ----- Waiting M211; ----- Advanced superimposition mode off. M103P12; ----- Waiting	~ M102P12; ----- Waiting M61; ----- Cancels superimposed control. M213; ----- Advanced superimposition mode off. M103P12; ----- Waiting
~	G31;
M104P12; ----- Waiting M210; ----- Advanced superimposition mode on. M105P12; ----- Waiting ~	M104P12; ----- Waiting M212; ----- Advanced superimposition mode on. M60; ----- Starts superimposed control. M105P12; ----- Waiting ~
: ------ Master axis move commands	~
~ M106P12; ----- Waiting M213; ----- Advanced superimposition mode off. M107P12; ----- Waiting	~ M106P12; ----- Waiting M61; ----- Cancels superimposed control. M213; ----- Advanced superimposition mode off. M107P12; ----- Waiting

NOTE

1. Start and cancel superimposed control for the slave axis. Specify the M code for starting or canceling superimposed control in a block without specifying other commands.
2. Use an unbuffered M code as the M code for turning the advanced superimposition mode on or off. (parameters Nos. 3411 to 3420)
3. Do not specify any move or PMC axis control command between the M code for turning the advanced superimposition mode on or off and a waiting M code.

Sequence of advanced superimposition and superimposed control start command

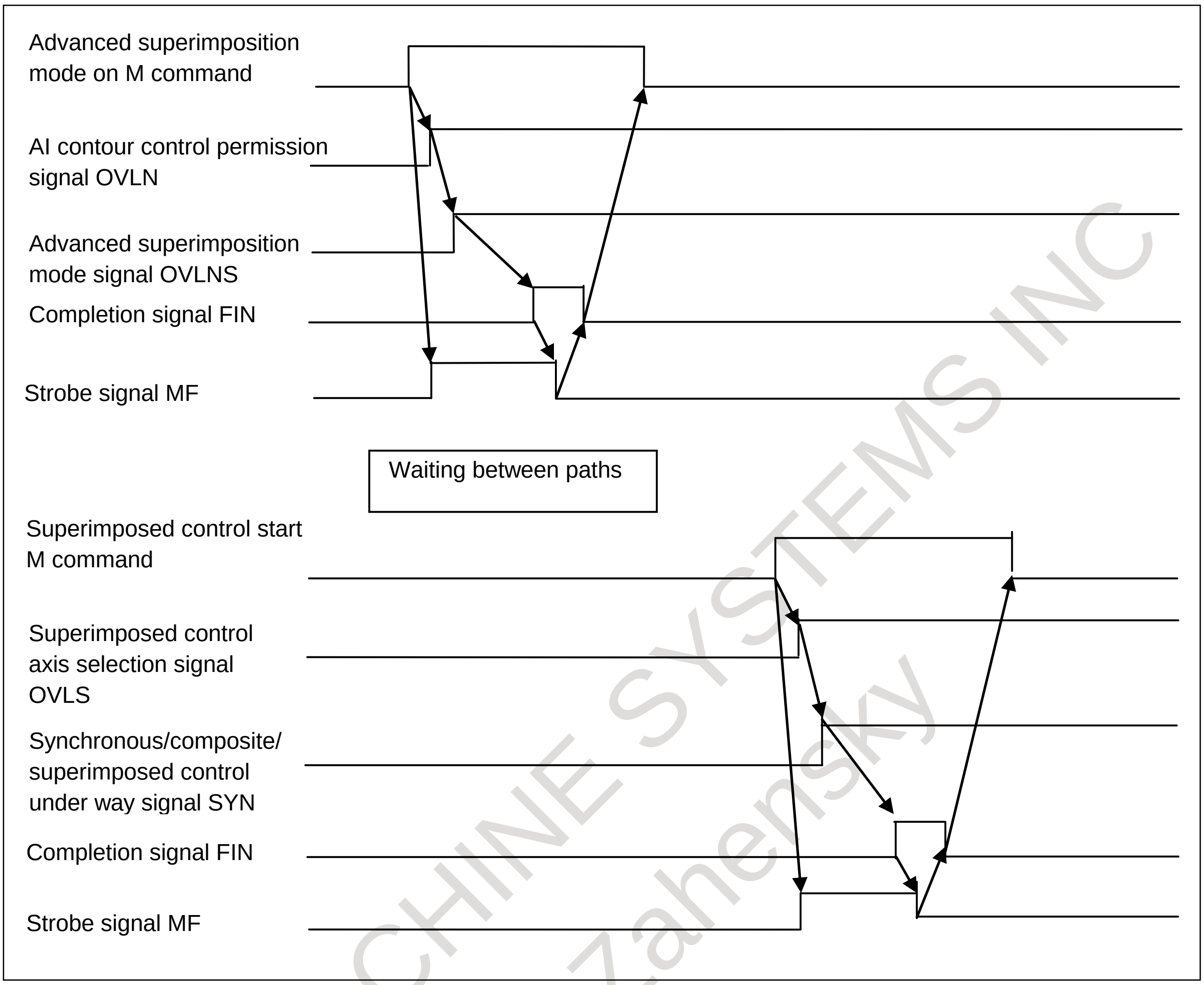


Fig. 8.10 (a)

1. When the M code for turning the advanced superimposition mode on is specified and the AI contour control permission signal OVLN is set to 1, the advanced superimposition mode signal OVLNS becomes 1.
2. The completion signal FIN is operated for completion.
3. The waiting M code is specified in both the master and slave axis paths.
4. When the M code for starting superimposed control is specified, and the superimposed control axis selection signal OVLS is set to 1, superimposed control starts. And, the superimposed control under way signal SYN becomes 1.
5. When the superimposed control under way signal SYN becomes 1, the completion signal FIN is operated for completion.

Sequence of advanced superimposition and superimposed control cancel command

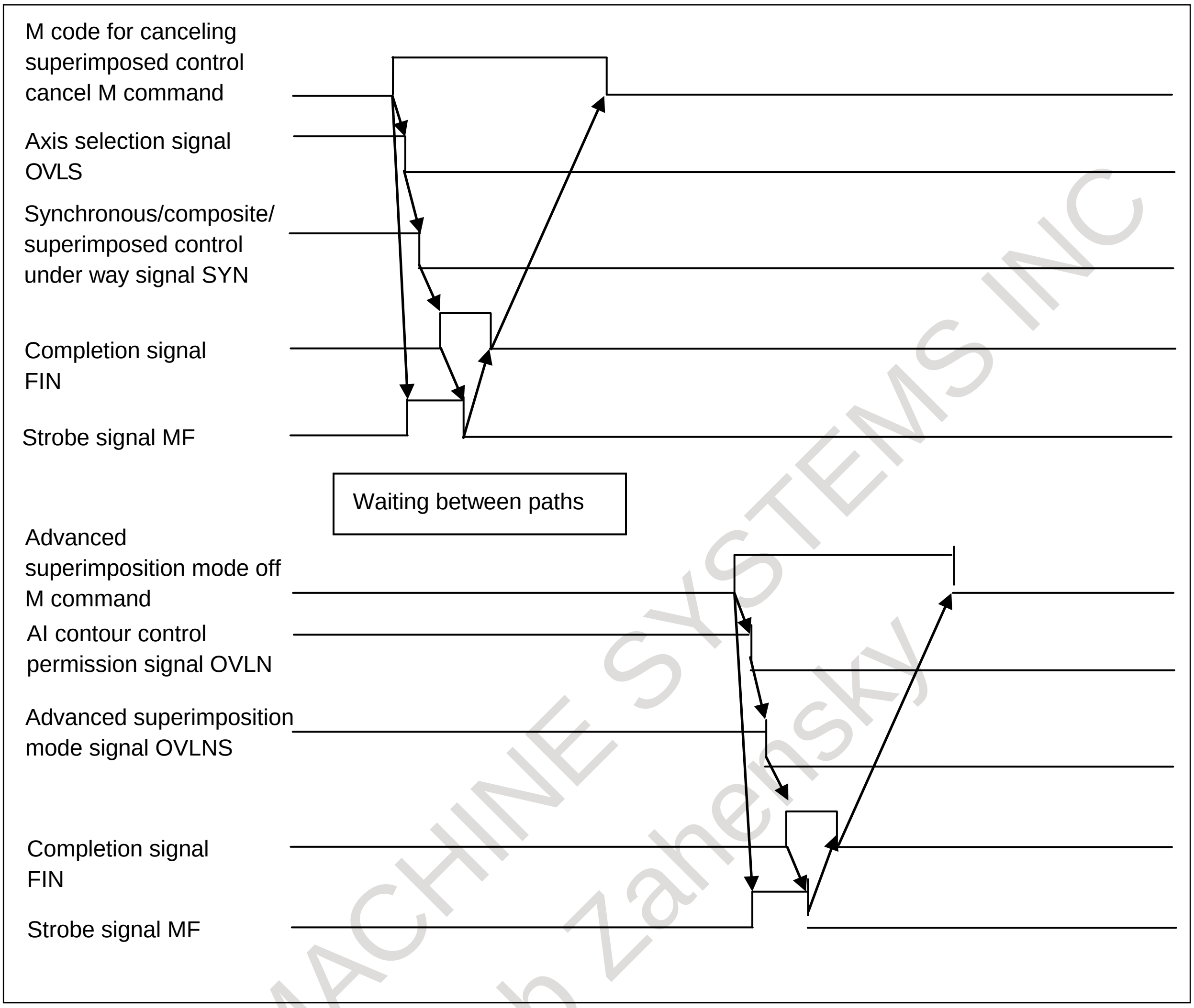


Fig. 8.10 (b)

1. When the M code for canceling superimposed control is specified and the superimposed control axis selection signal OVLS is set to 0, superimposed control is canceled.
2. When the superimposed control axis selection signal becomes 0, the completion signal FIN is operated for completion.
3. The waiting M code is specified in both the master and slave axis paths.
4. When the M code for turning the advanced superimposition mode off is specified and the AI contour control permission signal OVLN is set to 0, the advanced superimposition mode signal OVLNS becomes 0.
5. When the AI contour control permission signal OVLN becomes 0, the completion signal FIN is operated for completion.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8004					G8C			

[Input type] Parameter input
[Data type] Bit path

#3 G8C In PMC axis control, advanced preview feed forward is :
0: Disabled.
1: Enabled.

Signal

Superimposed control axis selection signals OVLS1 to OVLS8 <Gn190>

[Classification] Input signal
[Function] These signals perform superimposed control.
[Operation] When one of these signals becomes 1, the control unit operates as follows:
- Starts superimposed control with using the corresponding axis as the slave axis. Uses an axis as the master axis following the setting in parameter No. 8186.

Synchronous/composite/superimposed control under way signals SYN1O to SYN8O <Fn118>

[Classification] Output signal
[Function] These signals indicate that each axis is in the synchronous, composite, or superimposed control mode.
[Output cond.] These signals become 1 under the following condition:
- The corresponding axis is in the superimposed control mode.

These signals become 0 under the following condition:
- The corresponding axis is not in the superimposed control mode.

AI contour control permission signal OVLN <Gn531.4>

[Classification] Input signal
[Function] This signal enables superimposed control in the AI contour control mode by advanced superimposition.
[Operation] When this signal becomes 1, the control unit operates as follows:
- The corresponding path enters the advanced superimposition mode.

When the AI contour control permission signal is changed to 1 or 0, the tool must stop along all axes (other than PMC axes) in the target path. If the tool moves along any axis, alarm DS0071 is issued.

Advanced superimposition mode signal OVLNS <Fn545.1>

[Classification] Output signal
[Function] This signal indicates the advanced superimposition mode.
[Output cond.] This signal becomes 1 under the following condition:
- The corresponding path is in the advanced superimposition mode.

This signal becomes 0 under the following condition:
- The corresponding path is not in the advanced superimposition mode.

Alarm and message

Number	Message	Description
PS0502	ILLEGAL G-CODE (SUPERIMPOSED AHEAD)	A G code unavailable in the advanced superimposition mode was specified.
DS0071	START OR RELEASE OF SUPERIMPOSED AHEAD CANNOT BE DONE	To start or cancel the advanced superimposition mode, the tool must be stopped along all axes.
DS0072	MANUAL REFERENCE RETURN CANNOT BE DONE	In the advanced superimposition mode, manual reference position return cannot be performed.

Limitation

- Manual reference position return

In the advanced superimposition mode, manual reference position return cannot be performed. If an attempt is made to perform manual reference position return in the advanced superimposition mode, alarm DS0072 is issued.

- PMC axis control

In a system in which this function (AI contour control permission signal OVLN <Gn531.4>) is used, to use PMC axis control, it is necessary to set bit 3 (G8C) of parameter No.8004 to 1. For details, refer to "ADVANCED PREVIEW FEED FORWARD FOR PMC AXIS CONTROL".

- Others

The limitations on superimposed control and AI contour control are applied to this function. For basic specifications of and limitations on superimposed control and AI contour control, see the explanation of superimposed control and AI contour control in this manual.

8.11 PATH SPINDLE CONTROL

Overview

This function allows a workpiece attached to one spindle to be machined simultaneously with two tool posts and each of two workpieces attached to each of two spindles to be machined simultaneously with each of two tool posts.

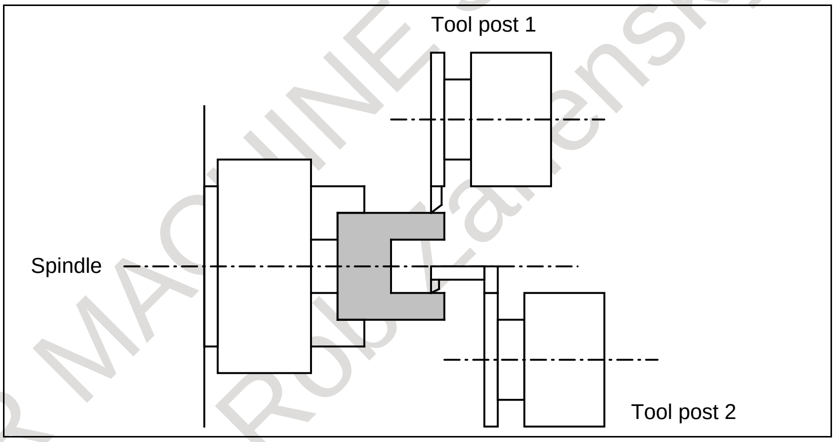


Fig. 8.11 (a) Application to a lathe with one spindle and two tool posts

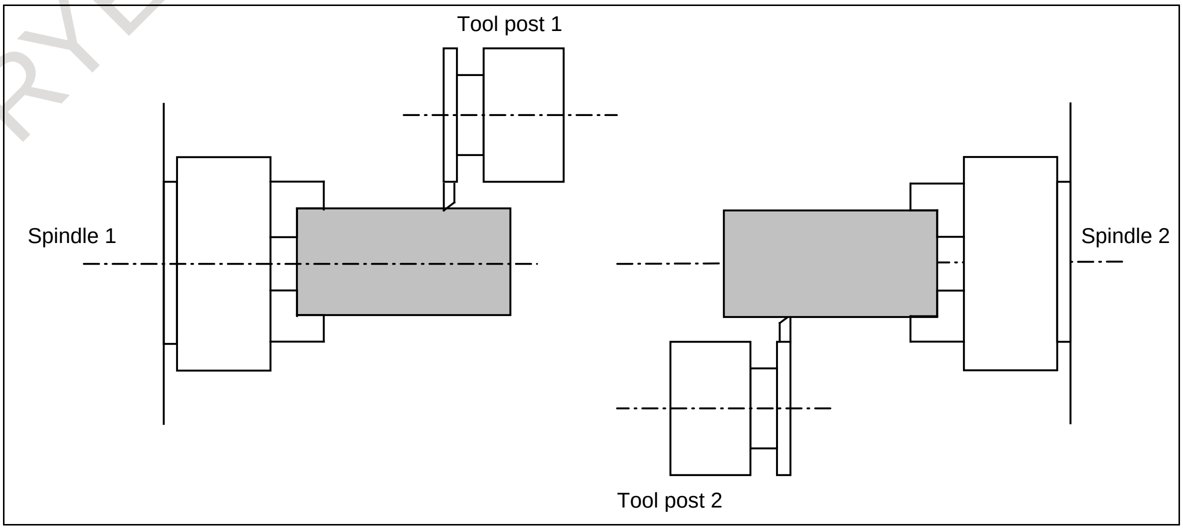


Fig. 8.11 (b) Application to a lathe with two spindles and two tool posts

The spindle belonging to each path can generally be controlled by programmed commands for the path. With path spindle command selection signals, programmed commands for any path can control the spindle belonging to any path.

When feed per revolution is performed on each path, the feedback pulses from the position coder attached on the spindle belonging to the path are generally used. With path spindle feedback selection signals, feed per revolution can be performed using the feedback pulses from the position coder attached on the spindle belonging to any path.

NOTE
1 The spindle speed is changed by spindle commands from multiple paths because path spindle command selection signals switch to spindle commands from relevant paths. Create a PMC sequence which determines whether spindle commands programmed for each path can be executed and switches over to another path spindle command selection signal.
2 When multiple paths share one analog spindle, during execution of a thread cutting command for a path, a function (such as thread cutting or feed per revolution) which uses the feedback pulses from the spindle on another path cannot be executed normally. Put restrictions so that during execution of thread cutting on a path, feed per revolution and other functions are not executed for the tool post on another path.

Explanation

- Selecting spindle commands
  Each path spindle command selection signal (input) can be used to switch over to spindle commands for the relevant path that the spindle belonging to each path follows.

- Selecting the feedback pulses from a position coder
  When a serial spindle is used, each path spindle feedback selection signal (input) can be used to switch over to the feedback pulses from the position coder attached on the spindle belonging to the relevant path that each path uses in the NC. Thread cutting and feed per revolution using the spindle belonging to a different path can be performed.

The following two signal types are available as selection methods using signals and each type is specified by parameter setting as follows.

Bit 0 (2P2) of parameter No. 3703	Signal type
0	Signal type A
1	Signal type B

The using method and meaning of signals differ depending on the signal type. For details, see the explanation of signals.

Each signal type is applicable to the following spindle configurations.

Signal type	Controlling the spindle belonging to path 1 from path 1 or 2	Controlling the spindle belonging to path 1 or 2 from path 1 or 2
Signal type A	○	×
Signal type B	○	○

> **NOTE**
> When the path spindle control function is used, the spindle gear selection method must be the T type.
> For a machining center system, set bit 4 (GTT) of parameter No. 3706 to 1 to select the T type as the spindle gear selection method.

- When multiple spindles belong to individual paths

Multi-spindle control can be used simultaneously to issue spindle commands to any spindle belonging to individual paths.
When multi-spindle control is not used, path spindle control is valid only for the 1st spindle belonging to individual paths.

The following two methods are available for selecting spindle commands in multi-spindle control:
1) Selecting spindle commands using signals (Bit 3 (MPP) of parameter No. 3703 is set to 0.)
2) Selecting spindle commands using address P (Bit 3 (MPP) of parameter No. 3703 is set to 1.)

When spindle commands are selected using address P, processing including path spindle command selection is performed in a multipath system. For this reason, spindle command selection using signals is invalid (this selection is made by spindle selection signals SWS1 to SWS4 <Gn027.0 to Gn027.2 and Gn026.3> and path spindle command selection signals SLSPA <Gn063.2>, SLSPB <Gn063.3>, SLSPC <Gn404.0>, and SLSPD <Gn404.1>).
Only one address P can be specified together with a spindle command and there is a one-to-one correspondence between address P values and spindles. For this reason, a spindle command in the same block cannot control multiple spindles simultaneously.

Selecting the feedback pulses from a position coder
The simultaneous use of multi-spindle control enables feed per revolution using the feedback pulses from the position coder on any spindle belonging to each path. Use signals (combination of position coder selection signal and path spindle feedback selection signal) to select the feedback pulses from a position coder.

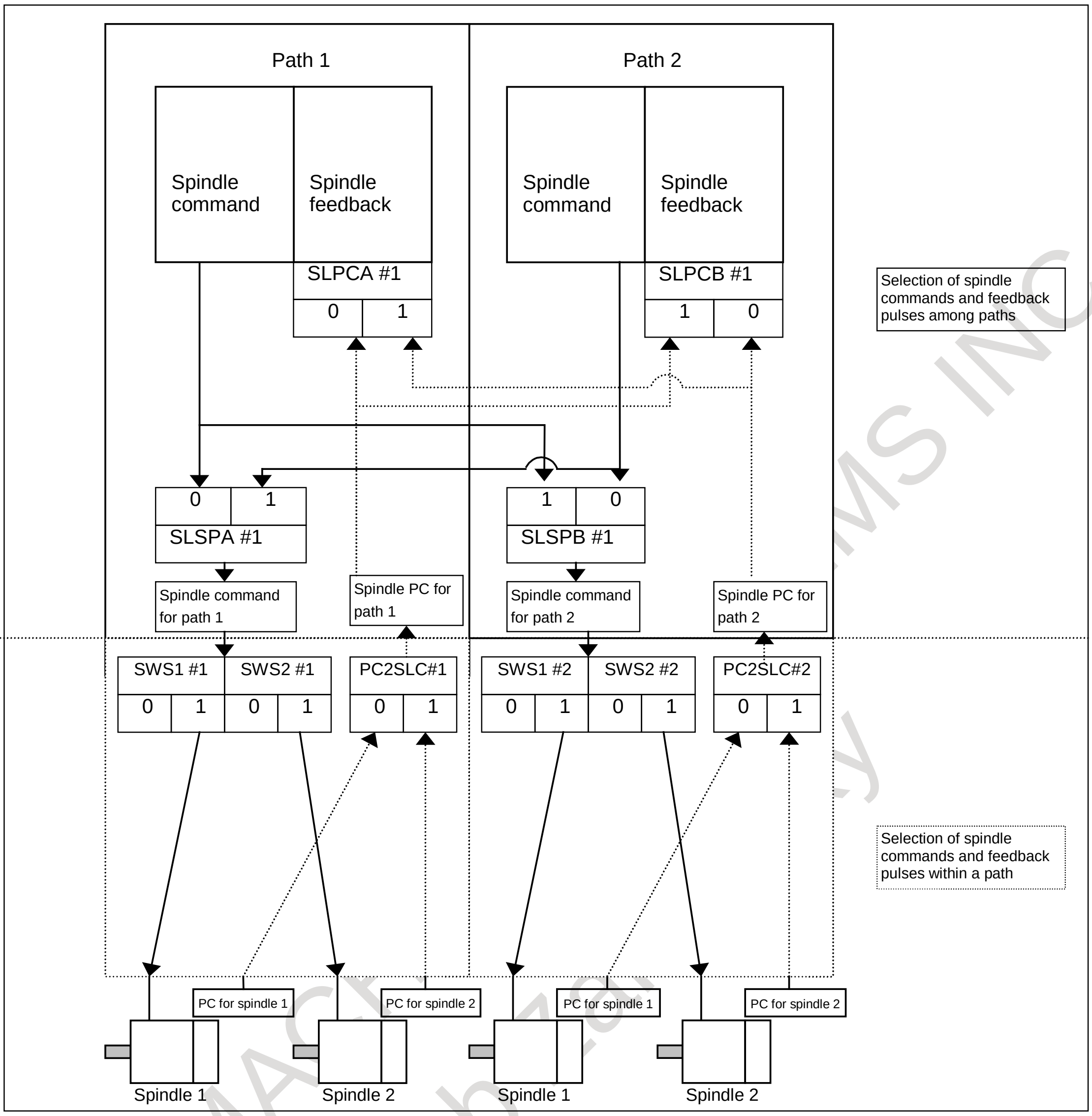


Fig. 8.11 (c) Configuration in which four spindles belonging to paths 1 and 2 are shared between the two paths (Path spindle control (signal type B) + multi-spindle control)

Signal

There are three signal types, which are selected with appropriate parameter settings.

Bit 2 (MPM) of parameter No. 3703	Bit 0 (2P2) of parameter No. 3703	Signal type
0	0	Signal type A
0	1	Signal type B
1	0/1	Signal type C

- Signal type A

Path spindle command selection signal SLSPA <G0063.2>

[Classification] Input signal

[Function] This signal specifies the path of the program commands to be effective to the spindles belonging to path 1.

[Operation]

Signal input SLSPA<G0063.2>	Program command to be effective to the spindle belonging to path m (= n + 1)
0	Spindle command of path 1
1	Spindle command of path 2

Path spindle feedback selection signal SLPCA<G0064.2>,SLPCB<G0064.3>

[Function] You need not control the path spindle feedback selection signal because the spindle feedback signal of path 1 is always effective to paths 1 and 2.

Path spindle command confirmation signal COSP <F0064.5>

[Classification] Output signal

[Function] This signal notifies which path the spindle command last specified came from.

[Output cond.] This signal becomes 1 under the following condition:

- A spindle command is issued from path 2.

This signal becomes 0 under the following condition:

- A spindle command is issued from path 1, or no spindle command is issued from either path.

In the state in which spindle commands for both paths 1 and 2 are effective to the spindles belonging to path 1, this signal allows you to determine which path the spindle command last specified came from.

> **NOTE**
> Spindle commands refer to S code commands, maximum speed command (G50S), M03/M04/M05, and commands for the constant surface speed control (G96 and G97).

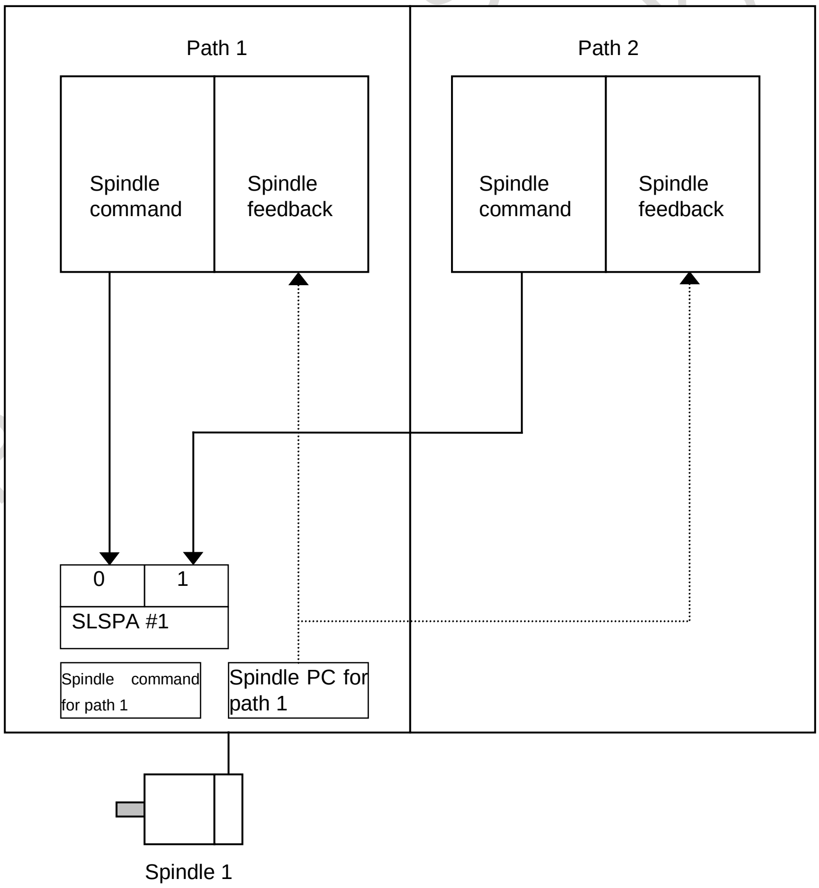


Fig. 8.11 (d) Configuration in which the spindle of path 1 is shared by 2 paths

- Signal type B

Path spindle command selection signal SLSPA<G0063.2>,SLSPB<G0063.3>

[Classification] Input signal

[Function] This signal specifies the path of the program commands to be effective to the spindles belonging to paths 1 and 2.

[Operation]

Signal input	Program command to be effective to the spindle belonging to path 1
SLSPA<G0063.2>	
0	Spindle command of path 1
1	Spindle command of path 2

Signal input	Program command to be effective to the spindle belonging to path 2
SLSPB<G0063.3>	
0	Spindle command of path 2
1	Spindle command of path 1

Path spindle feedback selection signal SLPCA<G0064.2>,SLPCB<G0064.3>

[Classification] Input signal

[Function] In paths 1 and 2, this signal specifies the path to which the spindle to which position coder feedback pulses are to be effective belongs.

[Operation]

Signal input	In path 1, the path to which the spindle to which position coder feedback pulses are to be effective belongs
SLPCA<G0064.2>	
0	Spindle PC of path 1
1	Spindle PC of path 2

Signal input	In path 2, the path to which the spindle to which position coder feedback pulses are to be effective belongs
SLPCB<G0064.3>	
0	Spindle PC of path 2
1	Spindle PC of path 1

Path spindle command confirmation signal COSP<F0064.5>

[Classification] Output signal

[Function] This signal notifies which path the spindle command last specified came from.

[Output cond.] This signal becomes 1 under the following condition:

- A spindle command is issued from path 2.

This signal becomes 0 under the following condition:

- A spindle command is issued from path 1, or no spindle command is issued from either path.

In the state in which spindle commands for both paths 1 and 2 are effective to the spindles belonging to path 1, this signal allows you to determine which path the spindle command last specified came from.

> **NOTE**
> Spindle commands refer to S code commands, maximum speed command (G50S), M03/M04/M05, and commands for the constant surface speed control (G96 and G97).

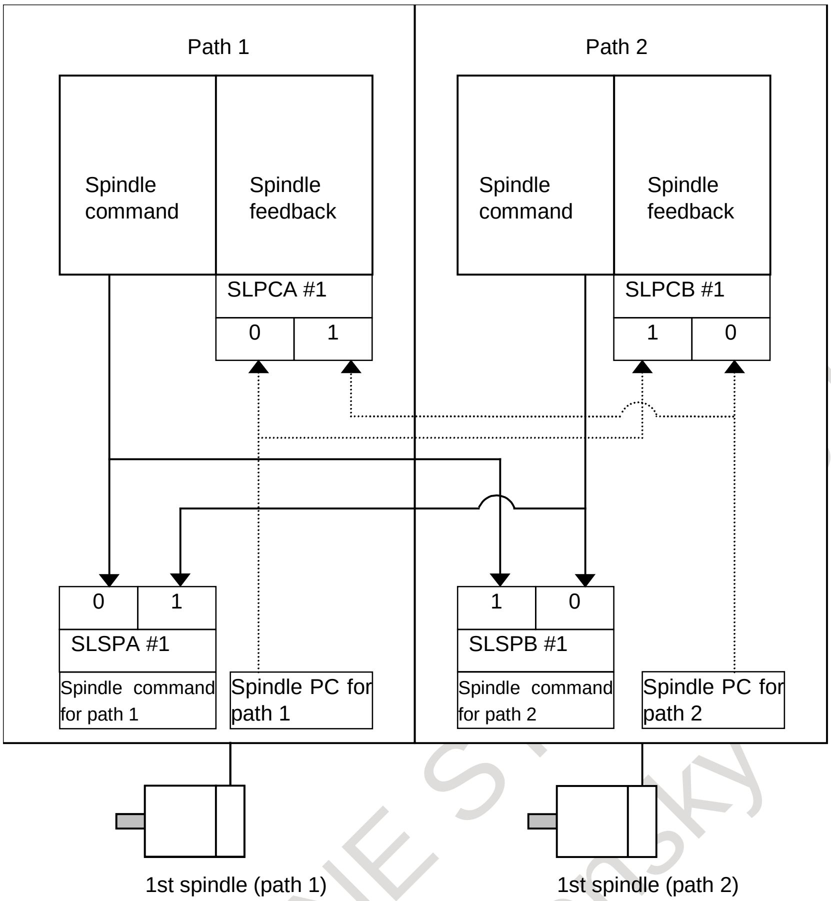


Fig. 8.11 (e) Configuration in which the spindles in paths 1 and 2 are shared by 2 paths

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn063					SLSPB	SLSPA		
Gn064					SLPCB	SLPCA		

	#7	#6	#5	#4	#3	#2	#1	#0
Fn064			COSP					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3702							EMS	

[Input type] Parameter input
[Data type] Bit path

#1 EMS The multi-spindle control is:
0: Used.
1: Not used.

NOTE
1 Set this parameter for a path that does not require multi-spindle control in a 2-path control system.
2 To use this parameter requires enabling multi-spindle control (setting bit 3 (MSP) of parameter No. 8133).

	#7	#6	#5	#4	#3	#2	#1	#0
3703					**MPP**	**MPM**		**2P2**

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 2P2 When a multi-path system is used, inter-path spindle control allows:
0: Configuration where the spindle that belongs to one path only is shared between path 1 and path 2.
1: Configuration where the spindles that belong to path 1 and 2 are shared between the two paths.

#3 MPP In multi-spindle control, a spindle selection using a programmed command instead of using Spindle selection signals (SWS1 to SWS4<G027.0 to 2, G026.3>) is:
0: Not made.
1: Made.

> **NOTE**
> When this parameter is set to 1, set parameter No. 3781 at the same time.

	#7	#6	#5	#4	#3	#2	#1	#0
3706					**PCS**			

[Input type] Parameter input
[Data type] Bit path

#3 PCS When a multi-path system is used, and multi-spindle control is enabled with each path, as the position coder signals (PC2SLC<Gn028.7>, PC3SLC<Gn026.0>, PC4SLC<Gn026.1>) for selecting the position coder of a spindle among the multiple spindles that belong to a path selected by the inter-path spindle feedback selection signals:
0: The signals of the path selected by the inter-path spindle feedback selection signal are used.
1: The signals of the local path are used.

Suppose that path x is selected by the inter-path spindle feedback selection signals (SLPCA<Gn064.2>, SLPCB<Gn064.3>, SLPCC<Gn403.4>, SLPCD<Gn403.5>). Then, the following position coder is selected in path x by the position coder selection signals:
n = m(path number)-1
y = x(path number selected by the spindle feedback selection signals)-1

When bit 3 (PCS) of parameter No. 3706 is set to 0

Position coder selected in path m	Selected path Position coder selection signals (path x)			Selecting path Position coder selection signals (path m)		
	PC2SLC <Gy028.7>	PC3SLC <Gy026.0>	PC4SLC <Gy026.1>	PC2SLC <Gn028.7>	PC3SLC <Gn026.0>	PC4SLC <Gn026.1>
PC1 of path x	0	0	0	-	-	-
PC2 of path x	1	0	0	-	-	-
PC3 of path x	0	1	0	-	-	-

Position coder selected in path m	Selected path Position coder selection signals (path x)			Selecting path Position coder selection signals (path m)		
	PC2SLC <Gy028.7>	PC3SLC <Gy026.0>	PC4SLC <Gy026.1>	PC2SLC <Gn028.7>	PC3SLC <Gn026.0>	PC4SLC <Gn026.1>
PC4 of path x	0	0	1	-	-	-

When bit 3 (PCS) of parameter No. 3706 is set to 1

Position coder selected in path m	Selected path Position coder selection signals (path x)			Selecting path Position coder selection signals (path m)		
	PC2SLC <Gy028.7>	PC3SLC <Gy026.0>	PC4SLC <Gy026.1>	PC2SLC <Gn028.7>	PC3SLC <Gn026.0>	PC4SLC <Gn026.1>
PC1 of path x	-	-	-	0	0	0
PC2 of path x	-	-	-	1	0	0
PC3 of path x	-	-	-	0	1	0
PC4 of path x	-	-	-	0	0	1

3781	P code for selecting the spindle in multi-spindle control

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767

If bit 3 (MPP) of parameter No. 3703 is set to 1, set the P code to select each spindle under multi-spindle control. Specify the P code in a block containing the S command.

[Example] If the P code value for selecting the second spindle is set to 3,
S1000 P3;
causes the second spindle to rotate at S1000.

NOTE
1 This parameter is valid if bit 3 (MPP) of parameter No. 3703 is set to 1.
2 If this parameter is set to 0, the corresponding spindle cannot be selected by a P code.
3 Under multi-path control, the P code specified here is valid for each path.
For instance, if the P code to select the first spindle of path 2 is set to 21, specifying S1000 P21; in path 1 causes the first spindle of path 2 to be rotated at S1000.
4 Identical P code values cannot be used for different spindles. (Identical P code values cannot be used even if the paths are different.)
5 When this parameter is used (when bit 3 (MPP) of parameter No. 3703 is set to 1), the spindle command selection signal is invalid.
6 To use this parameter, enable multi-spindle control (bit 3 (MSP) of parameter No. 8133 is 1).

	#7	#6	#5	#4	#3	#2	#1	#0
8133					MSP			

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#3 MSP Multi-spindle is
0: Not Used.
1: Used.

Alarm and message

Number	Message	Description
PS5305	ILLEGAL SPINDLE NUMBER	In a spindle select function by address P for a multiple spindle control, 1) Address P is not specified. 2) Parameter No. 3781 is not specified to the spindle to be selected. 3) An illegal G code which cannot be commanded with an S_P_; command is specified. 4) The multi spindle cannot be used because the bit 1 (EMS) of parameter No. 3702 is 1. 5) The spindle amplifier number of each spindle is not set in parameter No. 3717. 6) A prohibited command for a spindle was issued (parameter No. 11090). 7) An invalid value is set in parameter No. 11090.

Note

> **NOTE**
> Signals used to operate the spindle control unit are not influenced by path spindle command selection signals; process them with a PMC Ladder program, if required.
> (Example: SFRA<G0070.5> is always a forward rotation command to the first serial spindle control amplifier of path 1.)

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	SPINDLE CONTROL BETWEEN EACH PATH

8.12 MEMORY COMMON TO PATHS

Overview

In a multipath system, this function enables data within the specified range to be accessed as data common to all paths. The data includes tool compensation memory and custom macro common variables.

Explanation

- Tool compensation memory

Part or all of tool compensation memory for individual paths can be used as common data by setting parameter No. 5029.

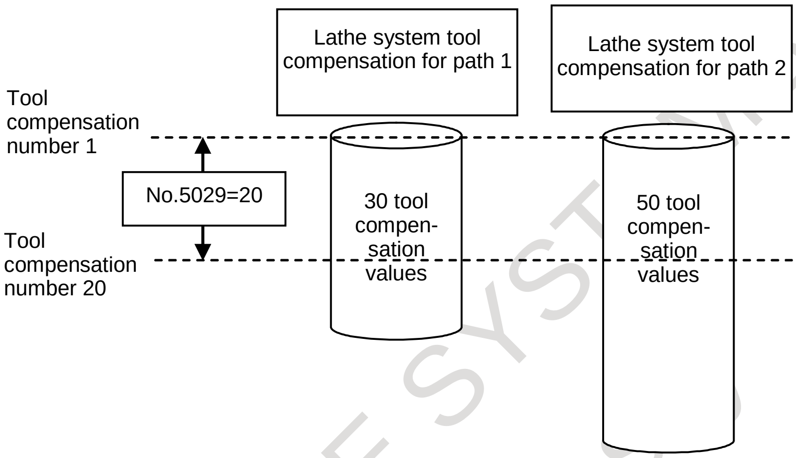


Fig. 8.12 (a)

> **NOTE**
> 1 Set a value less than the number of tool compensation values for each path for parameter No. 5029.
> 2 If the value set for parameter No. 5029 is greater than the number of tool compensation values for each path, the minimum number of tool compensation values for individual paths is assumed.

- Custom macro common variables

All or part of custom macro common variables #100 to #149 (or #199) and #500 to #599 (or #999) can be used as common data by setting parameters Nos. 6036 (#100 to #149 (, #199, or #499)) and 6037 (#500 to #599 (or #999)).

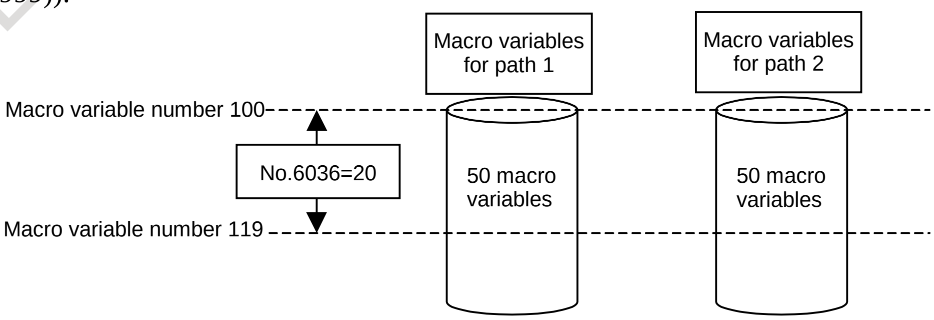


Fig. 8.12 (b)

> **NOTE**
> If the value of parameter No. 6036 or 6037 exceeds the maximum number of macro common variables, the maximum number of macro common variables is assumed.

Parameter

5024	Number of tool compensation values

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 999

Set the maximum allowable number of tool compensation values used for each path.
Ensure that the total number of values set in parameter No. 5024 for the individual paths is within the number of compensation values usable in the entire system. The number of compensation values usable in the entire system depends on the option configuration.
If the total number of values set in parameter No. 5024 for the individual paths exceeds the number of compensation values usable in the entire system, or 0 is set in parameter No. 5024 for all paths, the number of compensation values usable for each path is a value obtained by dividing the number of compensation values usable in the entire system by the number of paths.
Tool compensation values as many as the number of compensation values used for each path are displayed on the screen. If tool compensation numbers more than the number of compensation values usable for each path are specified, alarm PS0115 “VARIABLE NO. OUT OF RANGE” is issued.
For example, 64 tool compensation sets are used, 20 sets may be allocated to path 1, 30 sets to path 2, and 14 sets to path 3. All of 64 sets need not be used.

5029	Number of tool compensation value memories common to paths

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 999

When using memories common to paths, set the number of common tool compensation values in this parameter.
Ensure that the setting of this parameter does not exceed the number of tool compensation values set for each path (parameter No. 5024).

[Example 1] When parameter No. 5029 = 10, parameter No. 5024 (path 1) = 15, and parameter No. 5024 (path 2) = 30 in a 2-path system, tool compensation numbers 1 to 10 of all paths are made common.

[Example 2] When parameter No. 5029 = 20 and the other conditions are the same as for Example 1, tool compensation numbers 1 to 15 are made common.

> **NOTE**
> 1 Ensure that the setting of parameter No. 5029 does not exceed the number of tool compensation values for each path (parameter No. 5024). If the setting of parameter No. 5029 exceeds the number of compensation values of a path, the least of the numbers of compensation values in all paths is made common.

> **NOTE**
> 2 When 0 or a negative value is set, memories common to paths are not used.

6036	Number of custom macro variables common to tool path (for #100 to #199 (#499))

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 400

When the memory common to paths is used, this parameter sets the number of custom macro common variables to be shared (custom macro variables common to paths). Common variables #100 to #199 may be shared. Ensure that the maximum number of usable macro common variables is not exceeded.

[Example] When 20 is set in parameter No. 6036
#100 to #119: Shared by all paths
#120 to #149: Used by each path independently

> **NOTE**
> 1 To use from #160 to #199, enable Custom macro common variables(bit 6 (NCV) of parameter No.8135 is 0).
> 2 When 0 or a negative value is set, the memory common to paths is not used.

6037	Number of custom macro variables common to path (for #500 to #999)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 500

When the memory common to paths is used, this parameter sets the number of custom macro common variables to be shared (custom macro variables common to paths). Common variables #500 to #999 may be shared. Ensure that the maximum number of usable macro common variables is not exceeded.

[Example] When 50 is set in parameter No. 6037
#500 to #549: Shared by all paths
#550 to #599: Used by each path independently

> **NOTE**
> 1 To use from #600 to #999, enable Custom macro common variables(bit 6 (NCV) of parameter No.8135 is 0).
> 2 When 0 or a negative value is set, the memory common to paths is not used.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	COMMON MEMORY BETWEEN EACH PATH

8.13 PATH SINGLE BLOCK CHECK FUNCTION

Overview

In multi-path control, each of the paths has single block command signals, so that their respective automatic operation programs can be brought to a single block stop. Even if, however, the single block command signals of the respective paths are set to 1, the times at which the programs will stop may not match, depending on the difference among command programs.

Example

Assuming that the following programs are started to perform SBK operation at the position of X0.0 in both paths

Path 1
```
O0001;
N1 G01 X10.0 F100 ;
N2 X20.0 ;
   :
```

Path 2
```
O0002;
N1 G01 X20.0 F100 ;
   :
   :
```

In the above example, path 1 enters the single block stop state after the completion of the movement by "X10.0", while in path 2, the movement by "X20.0" will be executed immediately.
This function prevents such an event from occurring; as soon as one path enters the single block stop state, this function places other paths in the feedhold stop states. This function enables single block operation while making sure that the machining programs of multiple paths have nearly been synchronized.
Because the single block command signals of the individual paths remain effective, so that uses such as the following are possible:

<1> Single block operation in which the single block command signal of path 1 only is set to 1 to follow the program of path 1

<2> Single block operation in which the single block command signals of multiple paths are set to 1 to follow the end of a specified block in one of the paths.

⚠ WARNING

If one of the paths is in the single block prohibited state or in the feedhold prohibited state due to threading, etc, the paths will not enter the stop state until that state is canceled. However, the paths enter the stop state if the single block prohibited state is set by the custom macro system variable #3003.

⚠ CAUTION

No operations such as synchronous processing on paths are performed. In an example such as that shown above, therefore, path 2 enters the feedhold stop state after the completion of the movement by "X10.0", but the stop position will be near "X10.0".

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8100		DSB						

[Input type] Parameter input
[Data type] Bit machine group

#6 DSB The inter-path single block check function is:
0: Disabled.
When a single block stop occurs with a path, no single block stop occurs with the other path(s).
1: Enabled.
When a single block stop occurs with a path, a feed hold stop occurs with all paths in the same machine group.

8.14 PATH SELECTION/DISPLAY OF OPTIONAL PATH NAMES

Overview

Path selection specifies whether operations performed using the MDI unit is for each path.
The operations, as used here, include displaying and setting data items (such as tool compensation values), entering command programs in MDI mode, and editing machining programs in program memory.
Also note that the name of each path can be changed by parameter.

Signal

Path selection signals (Tool post selection signal)
HEAD<G063.0>

[Classification] Input signal
[Function] These signals select the path the MDI unit is to be for.
[Operation] Operations from the MDI unit will be on the path specified with HEAD. The relations between path selection signal combinations and selectable paths are as given in the table below.

Path selection signal HEAD G063.0	Path to be selected
0	Path 1
1	Path 2

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G063								HEAD

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8100							IAL	RST

[Input type] Parameter input
[Data type] Bit machine group

#0 RST The pressing of the 

key on the MDI unit is:
0: Valid for all paths in same machine group.
1: Valid only for the path selected by the path selection signal.

The  key on the MDI unit functions for all machine groups. So, in machine groups for which this parameter is set to 0, a reset can be performed for all paths. In machine groups for which this parameter is set to 1, a reset can be performed only for the path that is selected by the path select signal.

#1 IAL Choice of an option concerning operation continuation when an alarm is issued, and choice of an option concerning the start of automatic operation in alarm state:

0: When an alarm is issued, the operation is stopped with the other path(s) in same machine group placed in hold state.
When the other path or paths in same group are placed in alarm state, automatic operation cannot be started.

1: Even when an alarm is issued, the operation is continued without stopping the other path(s).
Even when the other path or paths in same group are placed in alarm state, automatic operation can be started.

3141	**Path name (1st character)**
3142	**Path name (2nd character)**
3143	**Path name (3rd character)**
3144	**Path name (4th character)**
3145	**Path name (5th character)**
3146	**Path name (6th character)**
3147	**Path name (7th character)**

[Input type] Parameter input
[Data type] Word path
[Valid data range] See the "CHARACTER CODE LIST".
Specify a path name with codes.
Any character string consisting of alphanumeric characters, katakana characters, and special characters with a maximum length of seven characters can be displayed as a series name.

> **NOTE**
> When 0 is set in parameter No. 3141, PATH1(,PATH2...) are displayed as path names.

CHARACTER CODE LIST

Character	Code	Comment	Character	Code	Comment	Character	Code	Comment
	032	Space	6	054		L	076	
!	033	Exclamation mark	7	055		M	077	
”	034	Quotation marks	8	056		N	078	
#	035	Sharp	9	057		O	079	
$	036	Dollar mark	:	058	Colon	P	080	
%	037	Percent	;	059	Semicolon	Q	081	
&	038	Ampersand	<	060	Left angle bracket	R	082	
’	039	Apostrophe	=	061	Equal sign	S	083	
(	040	Left parenthesis	>	062	Right angle bracket	T	084	
)	041	Right parenthesis	?	063	Question mark	U	085	
*	042	Asterisk	@	064	Commercial at mark	V	086	
+	043	Positive sign	A	065		W	087	
,	044	Comma	B	066		X	088	
-	045	Negative sign	C	067		Y	089	
.	046	Period	D	068		Z	090	
/	047	Slash	E	069		[	091	Left square bracket
0	048		F	070		¥	092	Yen mark
1	049		G	071		]	093	Right square bracket
2	050		H	072			094	
3	051		I	073		_	095	Underline
4	052		J	074				
5	053		K	075				

	#7	#6	#5	#4	#3	#2	#1	#0
3208			**PSC**					

[Input type] Setting input
[Data type] Bit

#5 PSC When the path is switched based on Path selection signal :
0: The screen display is switched to the last selected screen of the path.
1: The same screen as for the path before switching is displayed.

8.15 WAITING FUNCTION BY SPECIFYING START POINT

Overview

Control based on M codes is used to make one path to wait for the other during machining. When an M code for waiting is specified in a block of a path during automatic operation, the other path waits for the same M code to be specified before staring the execution of the next block.
In this function, by specifying a start point with a waiting M code, absolute coordinate value of that path or the other path can be used as the condition for waiting.

Format

M_ P3 L_ IP ;

Format to specify coordinates of a start point.
M : Waiting M code.
IP : Absolute coordinate value of a start point.
L : Specifying type of waiting (0,1).
L0 : This path waits until absolute coordinate on the other path arrives at start point.
L1 : The other path waits until absolute coordinate on this path arrives at start point.
P3 : Be sure to specify P3 (It can't be omitted).
∗The meaning of P command can be switched to "pattern of waiting with path number" by bit 1 (MWP) of parameter No. 8103.
For more detail information, please refer to "WAITING M CODES in this manual".

Note.
Specify absolute coordinate value of start point after M_ P_ L_. If start point is specified before M_ P_ L_, this block is operated as a normal waiting M code.
Specify L in an only one path.
If L is not specified in a block, this block is operated as a normal waiting M code, not the waiting function by specifying start point.

M_ P3 ;

Format not to specify coordinates of start point.

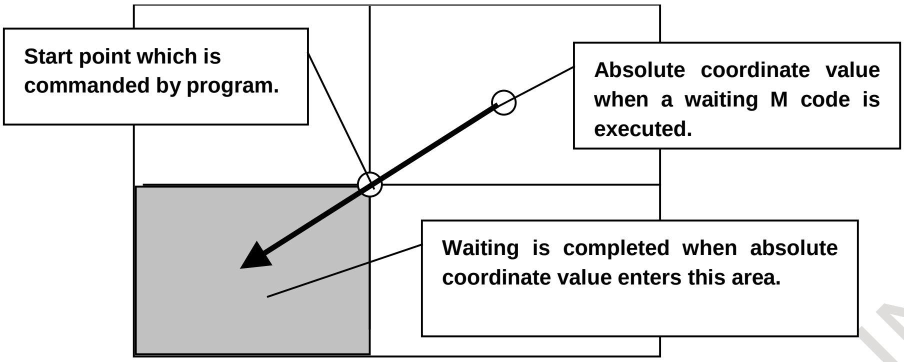
Operation

(1) When a waiting M code is specified on one path during automatic operation, that path waits until other paths execute the same waiting M code on the path with P command.

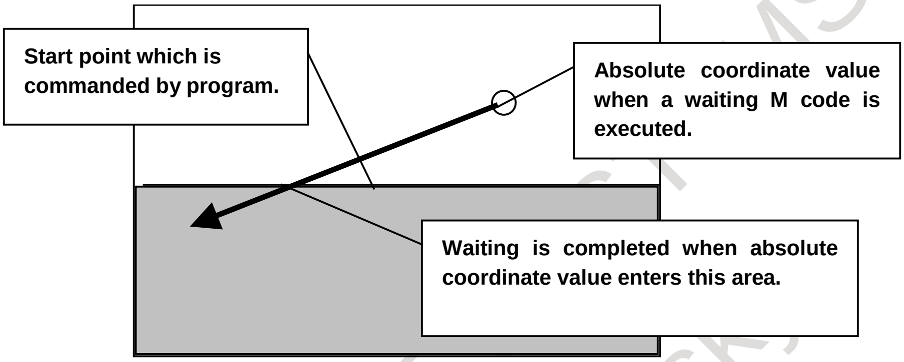
(2) After all related paths execute the same waiting M, The programmed coordinate values is compared with current ones, and a waiting area is decided.

Waiting is completed when absolute coordinate value enters the gray area in the following chart.

When command in coordinates for two axes



When command in coordinates for one axis



(3) If L0 is commanded, the other path executes a next block of a waiting M code, and the path with L0 waits until absolute coordinate value on the other path enters a gray area. If L1 is commanded, a next block of a waiting M code is executed, and the path with L1 makes the other path wait until absolute coordinate value of the path with L1 enters a gray area.

(4) After completing the waiting, the waited path begins to execute a following block of a waiting M code.

Example of program

L0 command

Path 1	Path 2
O0100; G00 X0.0 ; G00 Z0.0 ; : G00 X50.0 Z50.0 ;	O0100; G00 X0.0 Z0.0 ; M103 P3 L0 X10.0 Z10.0 ;
	Waiting for (1).
M103 P3;........................(1)	Path2 begins to wait till other path crosses coordinate values (X≤10.0, Z≤10.0).
X-30.0 Z-30.0 ;...............(2)	Path2 begins to execute a next block because coordinate values of other path crosses (X≤10.0, Z≤10.0) in the middle of block (2).

Path 1	Path 2
O0100; G00 X0.0 ; G00 Z0.0 ; : G00 X50.0 Z50.0 ; M103 P3;......................(1)	O0100; G00 X0.0 Z0.0 ; M103 P3 L0 X10.0 Z10.0 ;
	Waiting for (1)
	Path2 begins to wait till other path crosses coordinate values (X≤10.0, Z≤10.0).
S1000 M03 ; G99 G01 X-30.0 F0.2 ; ..(2) Z-30.0 ;.........................(3)	Path2 keeps waiting because the requirement of Z axis is not met though coordinate value of X axis crosses (X≤10.0) in block (2).
	Path2 begins to execute a next block because coordinate values crosses Z≤10.0 on the way of block (3).

L1 command

Path 1	Path 2
O0100; G00 X0.0 ; G00 Z0.0 ; : G00 X50.0 Z50.0 ; M103 P3;......................(1)	O0100; G00 X0.0 Z0.0 ; M103 P3 L1 X10.0 Z10.0 ; .... Waiting for (1)
Waiting till other path crosses coordinate values (X≥10.0, Z≥10.0)	Path2 begins to execute a next block
Path1 begins to execute a next block because coordinate values crosses (X≥10.0, Z≥10.0) in the middle of block (2).	G00 X50.0 Z50.0 ; ....(2)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3104	DAC	DAL						

[Input type] Parameter input
[Data type] Bit path

#6 DAL The displayed absolute position is
0: The actual position that takes tool length offset into account.
1: The programmed position that does not take tool length offset into account.

NOTE
In lathe systems, whether a tool offset is excluded from displaying absolute position is determined by the setting of bit 1 (DAP) of parameter No.3129.

#7 **DAC** When an absolute position are displayed:
0: Values not excluding the amount of travel based on cutter compensation and tool nose radius compensation are displayed.
1: Values excluding the amount of travel based on cutter compensation and tool nose radius compensation (programmed positions) are displayed.

NOTE
When the parameter DAC is set to 1, in the command like circle interpolation where the cutter compensation vector is changed constantly, an absolute position is not correctly displayed during its interpolation except its start point and its end point.

	#7	#6	#5	#4	#3	#2	#1	#0
3129							DAP	

[Input type] Parameter input
[Data type] Bit path

#1 DAP For absolute coordinate display:
0: The actual position including a tool offset (tool movement) is displayed.
1: The programmed position excluding a tool offset (tool movement) is displayed.

NOTE
In machining center systems, whether the tool length offset is excluded from displaying absolute position is determined according to the setting of bit 6 (DAL) of parameter No.3104

	#7	#6	#5	#4	#3	#2	#1	#0
8101							STW	

[Input type] Parameter input
[Data type] Bit path

#1 STW Waiting function by specifying start point is
0: Not effective.
1: Effective.

8110	Waiting M code range (minimum value)
8111	Waiting M code range (maximum value)

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 ,100 to 99999999
A range of M code values can be set by specifying a minimum waiting M code value (parameter No. 8110) and a maximum waiting M code value (parameter No. 8111).

(parameter No. 8110) ≤ (waiting M code) ≤ (parameter No. 8111)

Set 0 in these parameters when the waiting M code is not used.

Signal

No-wait signal
NMWT<Gn063.7>(for path individual signal interface)

[Classification] Input signal

[Function] Specifies whether to synchronize the paths by the waiting M code.

[Operation] When this signal turns to "1" the paths are not synchronized by the M code. The M code for waiting specified in a machining program is ignored.
When this signal turns to "0", the paths are synchronized by the M code. When the M code for waiting is specified for one path, the next block starts executed after the same M code is specified on another path.

> **NOTE**
> Set all signals NMWT <Gn063.7> to "1" for ignoring the waiting when using M code for the waiting function by specifying start point.

Waiting signal
WATO<F0063.6>(for path common signal interface)
WATO<Fn063.6>(for path individual signal interface)

[Classification] Output signal

[Function] Indicates that a waiting M code is being executed.

[Output cond.] These signals are "1" as long as:
- One path is waiting for another path. That is, the signal stays "1" for the period from when the M code for waiting is issued to one path to when the corresponding M code is issued to another path. In addition of the waiting function by specifying start point, the signal stays "1" when one path is waiting for the completion of the axis movement.

This signal is "0" as long as:
- Neither of the paths is waiting for the other.

Auxiliary function code signals M00 to M31<Fn010 to Fn013>

[Classification] Output signal

[Function] These signals report that auxiliary functions are specified.

[Output cond.] When the waiting function by specifying start point is available, these signals are output during waiting for other path M code or the completion of the axis movement.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn063	NWMT							
Fn010	M07	M06	M05	M04	M03	M02	M01	M00
Fn011	M15	M14	M13	M12	M11	M10	M09	M08
Fn012	M23	M22	M21	M20	M19	M18	M17	M16
Fn013	M31	M30	M29	M28	M27	M26	M25	M24
Fn063		WATO						

Alarm and message

Number	Message	Description
PS0003	TOO MANY DIGIT	Data entered with more digits than permitted in the NC instruction word. The number of permissible digits varies according to the function and the word.
PS0009	IMPROPER NC-ADDRESS	An illegal address was specified, or parameter 1020 is not set.
PS0160	MISMATCH WAITING M-CODE	A waiting M-code is in error. <1> When different M codes are specified for path 1 and path 2 as waiting M codes without a P command. <2> When the waiting M codes are not identical even though the P commands are identical <3> When the waiting M codes are identical and the P commands are not identical (This occurs when a P command is specified with binary value.) <4> When the number lists in the P commands contain a different number even though the waiting M codes are identical (This occurs when a P command is specified by combining path numbers.) <5> When a waiting M code without a P command (2-path waiting) and a waiting M code with a P command were specified at the same time <6> When the waiting function by specifying start point and a waiting M code without a P command (2-path waiting) were specified at the same time.
PS0161	ILLEGAL P OF WAITING M-CODE	P in a waiting M-code is incorrect. <1> When address P is negative <2> When a P value inappropriate for the system configuration was specified

Note

Inch / metric switching

Specifying coordinates of the waiting function by specifying start point might be failed when the input unit system is changed by G20/G21 during executing program.

```
O0100;
G20   ; (METRIC)
G00 X0 Z0. ;
  :
G21 ; (INCH)
G00 X50. Z50. ;
M103 P3;......................(1)
X-30. Z-30. ; .................(2)
```

```
O0100;
G00 X0. Z0. ;
M103 P3 L0 X10. Z10. ; .
(When M103 is executed,
the other path is a unit of
metric.)
Coordinates of a different
unit are compared because
the input unit system has
been changed when
waiting M code ends.
```

Current block display

The start position coordinates are not displayed in the current block display and the next block display of the program screen.

Execution macro

The waiting function by specifying start point cannot be used in execution macro (P-code program).
Also, macro cannot be called by M code of the waiting function by specifying start point.

Manual operation during waiting

Even if mode of operation is changed to the manual mode during waiting by specifying start position, waiting is completed when absolute coordinate value enters the specified area.
At this time, please change to JOG mode in only a path which axis moves, and please maintain memory mode in a path that is waiting.

Repetition of waiting command

When both of paths command L0, waiting is not completed and halt because each path waits for the movement of the other path.

Path 1)	Path2)
M_ P_ L0 X_ Z_ ; Halt	M_ P_ L0 X_ Z_ ; Halt

When a path command L1 and another path command L0, waiting of a path with L1 is completed and executes a next block, however waiting of a path with L0 is not completed and halt.

Path 1)	Path2)
M_ P_ L1 X_ Z_ ; Execute a next block	M_ P_ L0 X_ Z_ ; Halt

Amount of tool compensation and current position display

When the absolute coordinate value of the current position display reaches the start position, waiting is completed. Therefore, the start position can be shifted from the program coordinates when the tool compensation (wear-out compensation and tool-nose radius compensation etc.) is effective.

Example)

```
T0101 ; (When the amount wear-out compensation is X = -1.000)
G00 X0.0 ;
M103 P3 L1 X10.0 ;
G00 X20.0 ;
```

- When bit 6 (DAL) of parameter No. 3104 is 0, waiting is completed when absolute coordinate value is X = 10.000 and the amount of distance to go is X = 9.000.
- When bit 6 (DAL) of parameter No. 3104 is 0, waiting is completed when absolute coordinate value is X = 10.000 and the amount of distance to go is X = 10.000.

Manual handle retrace

Forward movement

There is not any restriction in forward movement.

Backward movement

When this function is effective, manual handle retrace function prohibits changing direction at a waiting M code block in backward movement. Continue backward movement until waiting M code is appeared in all paths.

> **NOTE**
> When this function or waiting M code is used with manual handle retrace, please set bit 4 (HMP) of parameter No. 6400 to 1 in the both of paths, and when inversion or backward movement is inhibited in other path, inversion or backward movement is inhibited also for the current executed path.

Flexible path axis assignment

Please do not execute flexible path axis assignment when paths are waiting.

Retrace (M system)

When the waiting function by specifying start point is available, a waiting M code cannot move backward. Only forward movement is possible.

Incremental command

Even if incremental command mode is selected by G91 command in G code system B or C at the path where M code for the waiting function by specifying start point is specified, it is assumed that absolute coordinate value is specified as a start point.
If address U, V or W, that are incremental commands, are specified as start point, the alarm PS0009 occurs.

Rotary axis roll- over

If a rotary axis which has roll-over function is specified as an axis of start point, this coordinate value, to which roll-over function is not applied, is compared with the current absolute coordinate value which is rounded by the angle corresponding to one rotation to judge whether a condition for waiting is met.

Example)
The axis A is the rotary axis and the amount of movement per rotation is 360.000
(Parameter No. 1260 = 360.000)

```
N10 G00 A0.0 ;
N20 M100 P3 L1 A180.0 ;
N30 G00 A-10.0 ;
```

After the end of N30 block, the absolute coordinate value of A axis is 350.0. Therefore, because this value becomes A≥180.0 in the middle of this block, waiting is completed.

8.16 FUNCTION FOR LOADER CONTROL

Overview

The function for loader control is used to control the devices for performing a non-machining operation (peripheral device such as a loader). When this function is valid, the path for performing a loader control is added besides a path for machining.
(In the subsequent explanation, the path which applies the function for loader control is called the loader path, while the other path is called the machining path.)

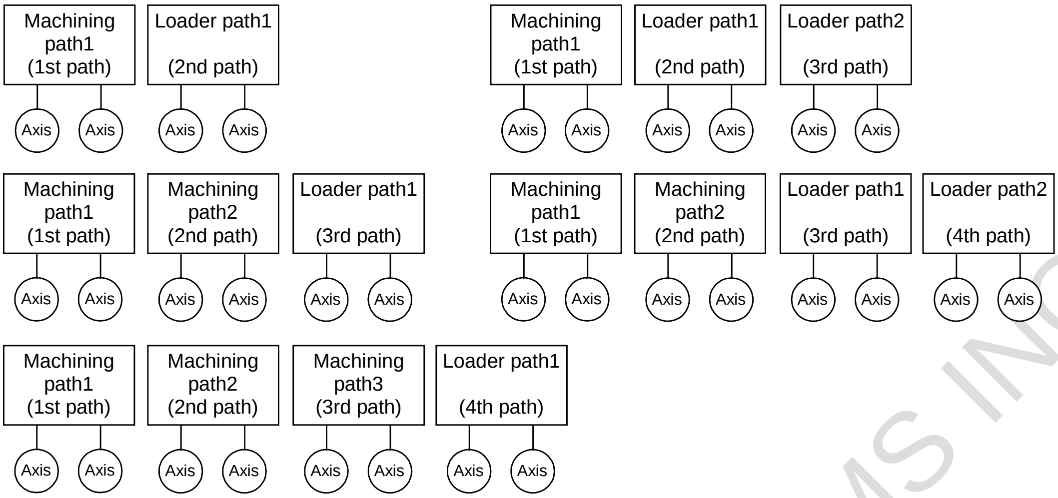
Explanation

- System configuration

When the function for loader control is valid, a combination of paths is as follows.

Machining path 1 + Loader path 1
Machining path 1 + Loader path 1 + Loader path 2
Machining path 1 + Machining path 2 + Loader path 1
Machining path 1 + Machining path 2 + Loader path 1 + Loader path 2
Machining path 1 + Machining path 2 + Machining path 3 + Loader path 2

Loader paths follow machining paths. When a number of machining paths is only 1, loader paths are added as the second or later path. When a number of machining paths is 2, loader paths are added as the third or later path. Loader path can be added up to 2 paths, but the total number of machining paths and loader paths cannot exceed 4.



> **NOTE**
> 1 To use the loader path 2, the addition of loader control path option is required in addition to the function for loader control option.
> 2 The function for loader control cannot be used in the system with more than 4 machining paths.

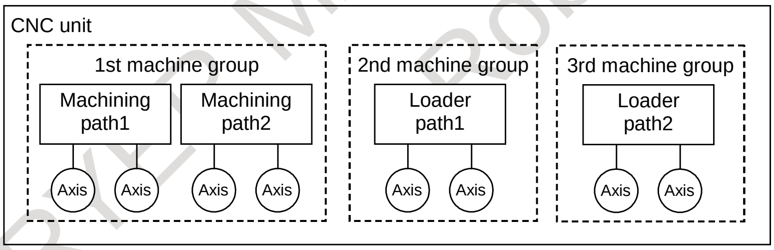
- Machine control type

When lathe system is specified as the machine control type, all paths including loader paths are controlled as a lathe. When machining center system is also specified as same, all paths are controlled as a machining center. When combined system is specified, each path can be selected either lathe system or machining center system. Select either type for a loader path.

- Machine group

A machine group is available on a loader path. Up to 3 groups are available, depending on the type of the CNC system.

Example)



- Number of feed axes

The maximum number of feed axes for the loader path is 4. However, when there are 2 loader paths, the maximum number of feed axes is 3 per path. When a controlled axis is assigned to a loader path, the absolute path number of a loader path is set to parameter No.981.

Example 1)

For a system with 2 paths and a total of 6 controlled axes:
Settings that cause the system to become a 2-path system with the machining path 1 consisting of 3 controlled axes and the loader path 1 consisting of 3 controlled axes.

Setting value of parameter No. 0981	Application
1	Machining path 1 (1st path), axis 1
1	Machining path 1 (1st path), axis 2

Setting value of parameter No. 0981	Application
1	Machining path 1 (1st path), axis 3
2	Loader path 1 (2nd path), axis 1
2	Loader path 1 (2nd path), axis 2
2	Loader path 1 (2nd path), axis 3

Example 2)

For a system with 4 paths and a total of 10 controlled axes:

Settings that cause the system to become a 4-path system with the machining path 1 consisting of 3 controlled axes, the machining path 2 consisting of 3 controlled axes, the loader path 1 consisting of 2 controlled axes and the loader path 2 consisting of 2 controlled axes.

Setting value of parameter No. 0981	Application
1	Machining path 1 (1st path), axis 1
1	Machining path 1 (1st path), axis 2
1	Machining path 1 (1st path), axis 3
2	Machining path 2 (2nd path), axis 1
2	Machining path 2 (2nd path), axis 2
2	Machining path 2 (2nd path), axis 3
3	Loader path 1 (3rd path), axis 1
3	Loader path 1 (3rd path), axis 2
4	Loader path 2 (4th path), axis 1
4	Loader path 2 (4th path), axis 2

- Designation of spindle axes

The loader path does not have a spindle. When the loader path is specified in a parameter No.982, which specifies the path to which each spindle belongs, alarm PS0365, “TOO MANY MAXIMUM SV/SP AXIS NUMBER PER PATH” is issued.

- Part program storage size / Number of registerable programs

A part program storage size and a number of registerable programs for the loader path are included in them of an entire system. Even when the loader path is added, part program storage size / number of registerable programs of an entire system do not change.

- DI/DO

The DI/DO specifications of the loader path confirm to the DI/DO specifications in multi-path control. In the loader path also, therefore, create a ladder program with the same interface as that for the machining path. Addresses of signals for a loader path are as follows.

- In case that a number of machining path is only 1:
  Loader path 1: F1000 to F1767, G1000 to G1767 (the 2nd path in the entire system)
  Loader path 2: F2000 to F2767, G2000 to G2767 (the 3rd path in the entire system)

- In case that a number of machining path is 2:
  Loader path 1: F2000 to F2767, G2000 to G2767 (the 3rd path in the entire system)
  Loader path 2: F3000 to F3767, G3000 to G3767 (the 4th path in the entire system)

- In case that a number of machining path is 3:
  Loader path 1: F3000 to F3767, G3000 to G3767 (the 4th path in the entire system)

- Path selection

When the loader control selection signal LCBS <G0251.1> turns to "1", the reference destination of screen display and MDI operation are switched to a loader path. If the loader path 2 exists, when the loader control 2 selection signal LCB2 <G0251.2> turns to "1", the loader path 2 is subjected to screen display or MDI operation.

When both signals are set to "1", the loader path 1 is selected.

When both the loader control selection signal and the loader control 2 selection signal are set to "0", a selected path can be switched between a machining path and a loader path by MDI key operation. Whenever MDI keys SHIFT and HELP are pressed at the same time, a selected path is switched between a machining path and a loader path alternately. If the loader path 2 exists, when the loader path 1 is selected, the loader path 2 is switched by the MDI key operation. In case of switching from a loader path to a machining path, a machining path which is selected by the path selection signals (HEAD <G0063.0>, H EAD2 <G0062.7>) is switched.

Example)
4-path system (machining path 1, machining path 2, loader path 1, and loader path 2)

- In case of selecting the machining path 1 by the path selection signal (HEAD="0", HEAD2="0") Whenever MDI keys SHIFT and HELP are pressed at the same time, a selected path is switched in the following order.

  Machining path 1 → Loader path 1 → Loader path 2

Path selection by MDI key operation can be disabled by setting bit 3 (SCD) of parameter No.8106.
When the loader path1 is selected, the loader path selection check signal LC1O<F0297.1> turns to "1".
When the loader path 2 is selected, the loader path 2 selection check signal LC2O<F0297.2> turns to "1".

When the loader path is selected, “LC” is displayed at the lower right of the screen. If the loader path 2 exists, “LC1” is displayed on the loader path 1 and “LC2” is displayed on the loader path 2. Optional path name can be displayed by setting parameters Nos. 3141 to 3147. A name of the loader path can be displayed in reverse video by setting bit 1 (LCI) of parameter No.11362.

- Waiting M codes

Waiting M codes are available between the loader path and the machining path. The following two methods for specifying paths which are to wait for one another at address P in the same block in which an M code for waiting is specified are available and can be selected using bit 1 (MWP) of parameter No. 8103. One method is to specify the paths with the sum of their corresponding binary values. The other is to specify them with their path numbers in combination.

- When bit 1 (MWP) of parameter No. 8103 is set to 0:

The sum of the binary values corresponding to the numbers of paths which are to wait for one another is specified with address P.

Path number	Binary value (decimal number)
1	1
2	2
3	4
4	8

Example)
4-path system (machining path 1, machining path 2, loader path 1, and loader path 2)
P5: Machining path 1 and loader path 1 wait for each other. (1+4)
P15: Machining path 1, 2 and loader path 1, 2 wait for one another. (1+2+4+8)

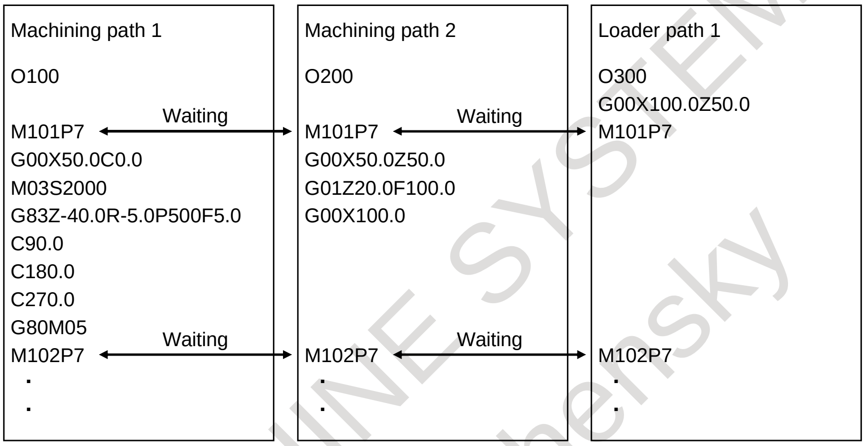
- When bit 1 (MWP) of parameter No. 8103 is set to 1:

A combination of numbers of all paths that are to wait for one another is specified with address P. If command values are different, when they consist of the same numeric characters, they are treated the same command.

Example)
4-path system (machining path 1, machining path 2, loader path 1, and loader path 2)

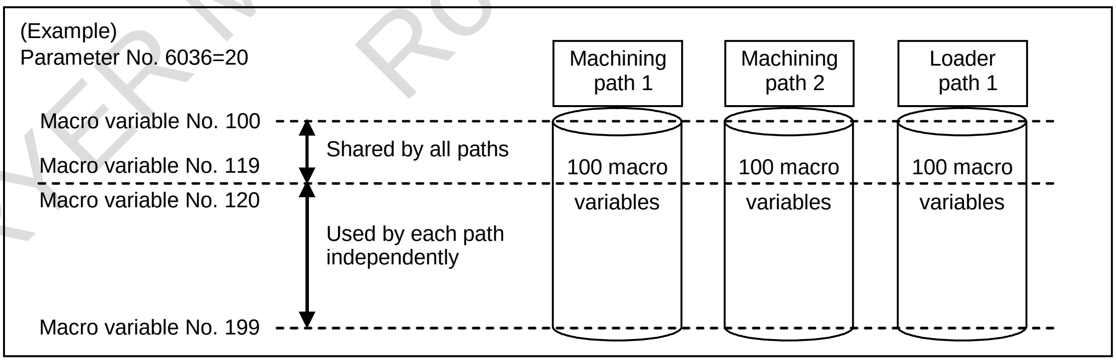
Making machining path 1 and loader path 1 wait for each other: the following 2 commands
P13, P31
Making machining path 1, 2 and loader path 1, 2 wait for one another: the following 24 commands
P1234, P1243, P1324, P1342, P1423, P1432, P2134, P2143, P2314, P2341, P2413, P2431
P3124, P3142, P3214, P3241, P3412, P3421, P4123, P4132, P4213, P4231, P4312, P4321

Example)
In case that M101 and M102 are waiting M codes and parameter MWP is set to 0:



- Custom macro common variables between each path

All or part of custom macro common variables #100 to #199 and #500 to #999 can be used as data common to all paths including the loader path by setting parameter Nos. 6036 (#100 to #199) and 6037 (#500 to #999).



- Position switch

Up to 24 position switch signals can be output in the loader path. 16 position switch signals (PSW01 to PSW16) are output to the conventional signal address. The other position switch signals (PSW17 to PSW24) are output to the address Fn330. The controlled axes corresponding to their position switches are specified in parameters Nos. 6966 to 6973, and the operating ranges are set in parameters Nos. 6974 to 6989. As to the machining path, up to 16 position switch signals is available conventionally.

> **NOTE**
> The position switch signals (PSW17 to PSW24) are available only for the loader path.

- Spindle speed control

The loader path does not have spindle speed control. If an S command is executed in the loader path, the specified value is output directly on the code signals as an auxiliary function.

- Tool functions

The loader path does not have tool functions. If a T command is executed in a loader path, the specified value is output directly on the code signals as an auxiliary function. The loader path does not have a tool offset number and the tool compensation screen.

- C language executer/Macro executer

When a loader path is specified by the C-language executor or the macro executor, use the following path number.

- In case that a number of machining path is only 1:
  Loader path 1: 2 (the 2nd path in the entire system)
  Loader path 2: 3 (the 3rd path in the entire system)

- In case that a number of machining path is 2:
  Loader path 1: 3 (the 3rd path in the entire system)
  Loader path 2: 4 (the 4th path in the entire system)

- In case that a number of machining path is 3:
  Loader path 1: 4 (the 4th path in the entire system)

- Reading of path information by FOCAS2/C language executer

The number of controlled paths (including loader paths) and path attribute can be read by cnc_sysinfo_ex function of the FOCAS2 library and the C language library of C language executor.

Please refer to the subsection of “the function specification” of FANUC FOCAS1/2 library specifications (FWLIB32.HTM) included in “FOCAS1/2 library disk” for details of cnc_sysinfo_ex function of the FOCAS2 library.

Please refer to the subsection of “the CNC/PMC window library” of C language Executor PROGRAMMING MANUAL (B-63943EN-3) for details of cnc_sysinfo_ex function of the C language library of C language executor.

- Notes

(1) In a system with a loader path, the following specifications are also the same as them for a multi-path control system.
  - Alarm
    If, during automatic operation, an alarm occurs in a certain path, all paths in the machine group to which the path belongs will enter the feedhold state and stop. However, the operations in other paths in the same machine group can be continued without generating an alarm, by appropriately setting bit 1 (IAL) of parameter No. 8100.
  - MDI reset key
    The reset key on the MDI unit is effective to all paths. Note, however, that the key can be enabled only for the path selected with the path selection signal, by setting bit 0 (RST) of parameter No. 8100. (The key is also effective for the loader path which is selected by the loader control selection signal or MDI key operation)

(2) Input signals by the X address are not assigned to the 4th path in an entire system. Therefore, when a loader path is the 4th path in an entire system, the signals must be assigned by setting bit 2 (XSG) of parameter No. 3008. If the parameter XSG is set to 0, alarm PW0001, "X-ADDRESS IS NOT ASSIGNED" is issued in the loader path.

- Usable G codes

The following table shows the G codes which are available for a loader path. When the G codes which are not listed in this table are specified in a loader path, alarm PS0010, "IMPRPOER G-CODE" is issued. G code system for a loader path can be set to a different system for a machining path. Which G code system is selected is set using bit 7 (GSC) and 6 (GSB) of parameter No. 3401.

Machining center system	Lathe system G code system			Function
	A	B	C	
G00	G00	G00	G00	Positioning (Rapid traverse)
G01	G01	G01	G01	Linear interpolation (Cutting feed)
G04	G04	G04	G04	Dwell
G04.1	G04.1	G04.1	G04.1	G code preventing buffering
G09	G09	G09	G09	Exact stop
G10	G10	G10	G10	Programmable data input
G11	G11	G11	G11	Programmable data input mode cancel
G20	G20	G20	G70	Input in inch
G21	G21	G21	G71	Input in mm
G22	G22	G22	G22	Stored stroke check function on
G23	G23	G23	G23	Stored stroke check function off
G27	G27	G27	G27	Reference position return check
G28	G28	G28	G28	Return to reference position
G30	G30	G30	G30	2nd, 3rd and 4th reference position return
G31	G31	G31	G31	Skip function
G92	G50	G92	G92	Coordinate system setting
G92.1	G50.3	G92.1	G92.1	Workpiece coordinate system preset
G52	G52	G52	G52	Local coordinate system setting
G53	G53	G53	G53	Machine coordinate system setting
G54	G54	G54	G54	Workpiece coordinate system 1 selection
G54.1	G54.1	G54.1	G54.1	Additional workpiece coordinate systems
G55	G55	G55	G55	Workpiece coordinate system 2 selection
G56	G56	G56	G56	Workpiece coordinate system 3 selection
G57	G57	G57	G57	Workpiece coordinate system 4 selection
G58	G58	G58	G58	Workpiece coordinate system 5 selection
G59	G59	G59	G59	Workpiece coordinate system 6 selection
G61	G61	G61	G61	Exact stop mode
G63	G63	G63	G63	Tapping mode
G64	G64	G64	G64	Cutting mode
G65	G65	G65	G65	Macro call
G66	G66	G66	G66	Macro modal call
G67	G67	G67	G67	Macro modal call cancel
G94	G98	G94	G94	Feed per minute
G90	-	G90	G90	Absolute programming
G91	-	G91	G91	Incremental programming

- Specifications

The following table shows functions which are available for a loader path. The functions which are not listed in this table are not available. Some of the following functions are optional functions. If required options are not specified, these functions cannot be used even on a loader path.

Item	Specifications
Controlled axis	
Max. feed axes/1 path	4 axes(1 path) / 3 axes(2 paths)
Axis control by PMC	
Axis name	
Axis name expansion	
Incremental system (IS-A, IS-B, IS-C)	
HRV2 control	
HRV3 control	
Inch/metric conversion	
Interlock	
Machine lock	
Emergency stop	
Overtravel	
Stored stroke check 1	
Stored stroke check 2,3	
Stroke limit check before move	
Mirror image	
Follow-up	
Servo off/Mechanical handle	
Unexpected disturbance torque detection function	
Position switch	
High speed position switch	
Dual check safety	
Test mode function for Acceptance Test	Dual check safety is required.
Operation	
Automatic operation (memory)	
MDI operation	
Program restart	
Quick program restart	
Manual intervention and return	
Wrong operation prevention	
Dry run	
Single block	
Jog feed	
Manual reference position return	
Manual 2nd/3rd/4th reference position return	
Reference position setting without DOG	
Reference position setting with mechanical stopper	
Reference point setting with mechanical stopper by Grid Method	
Manual handle feed	
High speed program check	
Interpolation functions	
Positioning	G00
Exact stop mode	G61
Tapping mode	G63
Cutting mode	G64
Exact stop	G09

Item	Specifications
Linear interpolation	G01
Dwell	G04, Dwell in seconds
Skip	G31
High-speed skip	
Torque limit skip	
Reference position return	G28
Reference position return check	G27
2nd reference position return	G30
3rd/4th reference position return	
Feed function	
Rapid traverse override	F0, 25, 50, 100% or 0～100%(1% Step)
Feed per minute	
Automatic acceleration/deceleration	
Rapid traverse bell-shaped acceleration/deceleration	
Linear acceleration/deceleration after cutting feed interpolation	
Bell-type acceleration/ deceleration after cutting feed interpolation	
Feedrate override	0～254%
One-digit F code feed	M series only
Jog override	0～655.34%
External deceleration	
Rapid traverse block overlap	
Programmable rapid traverse overlap	
Smart overlap	
In-position check signal	
Program input	
Optional block skip	9
Program file name	
Absolute/incremental programming	G90/G91
Decimal point programming/ pocket calculator type decimal point programming	
Diameter/radius programming	
Rotary axis designation	
Coordinate system setting	
Workpiece coordinate system	G52～G59
Addition of workpiece coordinate system	G54.1, M series only
Workpiece coordinate system preset	
G code system (A/B/C)	T series only
Programmable data input	G10
Programmable parameter input	
Sub program call	
Custom macro	
Custom macro common variables between each path	
G code preventing buffering	
Macro executor	
Macro executor + C language executor	
FANUC PICTURE executor	
FANUC PICTURE function	

Item	Specifications
Auxiliary function	
Auxiliary function	M8 digit
2nd auxiliary function	B8 digit
High-speed M/S/T/B interface	
Waiting function	
Waiting M codes of high-speed type	
Multiple command of auxiliary function	5 commands
Waiting function by specifying start point	
Accuracy compensation function	
Backlash compensation	
Backlash compensation for each rapid traverse and cutting feed	
Stored pitch error compensation	
Editing operation	
Part program editing	
Extended part program editing	
Key and program encryption	
Password function	
Playback	
Background editing	
Memory card program edit & operation	
Multi-path editing function	
Setting and display	
Optional path name display	
Operating monitor screen	
Servo setting screen	
Servo waveform display	
Trouble diagnosis	
Machine alarm diagnosis	
Software operator's panel	
Machine operation menu	
Multi-language display	Machining paths and loader paths are displayed in the same language.
Data protection key	
Protection of data at eight levels	
Warning function against modification of setting	
Graphic display	
Data input/output	
Reader/puncher interface	
External machine zero point shift	
External message	
External data input	
External key input	
External workpiece number search	
External program number search	
Memory card input/output	
USB memory input/output	
One touch macro call	

Signal

Path selection signals HEAD<G0063.0>, HEAD2<G0062.7>

[Classification] Input signal

[Function] These signals select the path the MDI unit is to be for.

[Operation] Operations from the MDI unit will be on the path specified with a combination of HEAD and HEAD2. The relations between path selection signal combinations and selectable paths are as given in the table below.

Path selection signal		Path to be selected
HEAD2<G0062.7>	HEAD<G0063.0>	
"0"	"0"	Path 1
"0"	"1"	Path 2
"1"	"0"	Path 3
"1"	"1"	Path 4

> **NOTE**
> The loader path cannot be selected by path selection signals.

Loader control selection signal LCBS<G0251.1>
Loader control 2 selection signal LCB2<G0251.2>

[Classification] Input signal

[Function] These signals switch the object of screen display or MDI operation between a machining path and a loader path.

[Operation] When LCBS turns to "1", the loader path 1 is subjected to screen display or MDI operation. When LCB2 turns to "1", the loader path 2 is subjected to screen display or MDI operation. When both signals are set to "1", the loader path 1 is selected.
When LCBS or LCB2 is set from "1" to "0", a machining path is switched. Selected path is a machining path which is selected by the path selection signals (HEAD<G0063.0>, HEAD2<G0062.7>).

> **NOTE**
> 1 LCB2 is available when loader path 2 is existed.
> 2 If this signal is set to “0”, a selected path can be switched between the machining path and the loader path by pressing MDI key [⇧ SHIFT] and [HELP] at the same time.

Loader path selection check signal LC1O<F0297.1>
Loader path 2 selection check signal LC2O<F0297.2>

[Classification] Output signal

[Function] These signals indicate that the object of screen display or MDI operation is a loader path.

[Output cond.] These signals are “1” in the following case:
- The object of screen display or MDI operation is the corresponding loader path.

These signals are “0” in the following case:
- The object of screen display or MDI operation is not the corresponding loader path.

LC1O is for the loader path 1, and LC2O for the loader path 2.

No-wait signal NOWT<G0063.1>(for path common signal interface) NMWT<Gn063.7>(for path individual signal interface)

[Classification] Input signal

[Function] Specifies whether to synchronize the paths by the waiting M code.

[Operation] When this signal turns to “1”, the paths are not synchronized by the M code. The M code for waiting specified in a machining program is ignored.

When this signal turns to “0”, the paths are synchronized by the M code. When the M code for waiting is specified for one of paths, wait the corresponding M code is commanded in another path, then starts to execute the next block.

Waiting signal WATO<F0063.6>(for path common signal interface) WATO<Fn063.6>(for path individual signal interface)

[Classification] Output signal

[Function] Indicates that the M code is waiting for a path.

[Output cond.] These signals are “1” as long as:

- One path is waiting for another path. That is, the signal stays “1” for the period from when the M code for waiting is issued to one path to when the corresponding M code is issued to another path.

This signal is “0” as long as:

- Neither of the paths is waiting for the other.

> **NOTE**
> When the path common signal interface (bit 0 (MWT) of parameter No. 8103 = 1) is used, F0063.6="1" results even if path 2 is placed in the wait state.
> So, when the wait state needs to be checked with each path, use the path individual signal interface (bit 0 (MWT) of parameter No. 8103 = 0).

Position switch signals PSW17 to PSW24<Fn330>

[Classification] Output signal

[Function] These signals indicate that the machine coordinates along the controlled axes specified by parameters Nos. 6966 to 6973 are within the ranges specified by parameters Nos. 6974 to 6989.
The position switch signal corresponding to the n-th position switch function is PSWn. (n : 17 to 24)

[Output cond.] These signals are “1” in the following case:

- When the machine coordinates along the controlled axes are within the specified ranges.

These signals are “0” in the following case:

- When the machine coordinates along the controlled axes are not within the specified ranges.

> **NOTE**
> This signal is available only for the loader path.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0062	HEAD2							
G0063								HEAD
G0251						LCB2	LCBS	
F0297						LC2O	LC1O	
Fn330	PSW24	PSW23	PSW22	PSW21	PSW20	PSW19	PSW18	PSW17

- For path common signal interface (bit0 (MWT) of parameter No.8103 =1)

	#7	#6	#5	#4	#3	#2	#1	#0
G0063							NOWT	
F0063		WATO						

- For path individual signal interface (bit0 (MWT) of parameter No.8103 =0)

	#7	#6	#5	#4	#3	#2	#1	#0
Gn063	NMWT							
Fn063		WATO						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0138								MDP

[Input type] Parameter input
[Data type] Bit

#0 MDP To the extensions of input/output files, a path number is:
0: Not added.
1: Added.

NOTE
The extension of the loader path 1 is “PL1” and the loader path 2 is “PL2”.

3131	Subscript of axis name

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 9, 65 to 90

In order to distinguish axes under synchronization control, and tandem control, specify a subscript for each axis name.

Setting value	Meaning
0	Each axis is set as an axis other than synchronization axis and tandem control axis.
1 to 9	A set value is used as a subscript.
65 to 90	A set letter (ASCII code) is used as a script.

[Example] When the axis name is X, a subscript is added as indicated below.

Setting value	Axis name displayed on a screen such as the position display screen
0	X
1	X1
77	XM
83	XS

If a multi-path system is used, no extended axis name is used within a path, and no subscript is set for the axis names, then the path number is automatically used as the subscript for the axis names. Note, however, a loader path name is as follows.

- In case that the loader path 2 does not exit: “L”
- In case that the loader path 2 exits: “L1”(loader path 1), “L2”(loader path 2)

To disable the display of axis name subscripts, set a blank (32) of ASCII code in the parameter for specifying an axis name subscript.

NOTE
If even one axis in a path uses an extended axis name when bit 2 (EAS) of parameter No. 11308 is set to 0, subscripts cannot be used for axis names in the path.

3141	Path name (1st character)
3142	Path name (2nd character)
3143	Path name (3rd character)
3144	Path name (4th character)
3145	Path name (5th character)
3146	Path name (6th character)
3147	Path name (7th character)

[Input type] Parameter input
[Data type] Word path
[Valid data range] See the character-code correspondence table.

Specify a path name with code.
Any character string consisting of alphanumeric characters, katakana characters and special characters with a maximum length of seven characters can be displayed as a series name.
When the parameter (No.3141) is set to 0 in a loader path, a path name is as follows.
- In case that the loader path 2 does not exit: “LC”
- In case that the loader path 2 exits: “LC1”(loader path 1), “LC2”(loader path 2)

	#7	#6	#5	#4	#3	#2	#1	#0
3401	GSC	GSB						

[Input type] Parameter input
[Data type] Bit path

#6 GSB The G code system is set.
#7 GSC

GSC	GSB	G code
0	0	G code system A
0	1	G code system B
1	0	G code system C

6036	Number of custom macro variables common to tool path (for #100 to #199)

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 100

When the memory common to paths is used, this parameter sets the number of custom macro common variables to be shared (custom macro variables common to paths).
Common variables #100 to #199 may be shared. Ensure that the maximum number of usable macro common variables is not exceeded.

> **NOTE**
> When 0 or a negative value is set, the memory common to paths is not used.

6037	Number of custom macro variables common to tool path (for #500 to #999)

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 500

When the memory common to paths is used, this parameter sets the number of custom macro common variables to be shared (custom macro variables common to paths).
Common variables #500 to #999 may be shared. Ensure that the maximum number of usable macro common variables is not exceeded.

> **NOTE**
> When 0 or a negative value is set, the memory common to paths is not used.

6966	Controlled axis for which the 17-th position switch function is performed
to	to
6973	Controlled axis for which the 24-th position switch function is performed

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the controlled axis number corresponding to one of the seventeenth to twenty-fourth position switch functions. When the machine coordinate of the corresponding axis is within a parameter-set range, the corresponding position switch signal is output to the PMC.

> **NOTE**
> 1 The setting of 0 means that the position switch function is not used.
> 2 The 17-th to 24-th position switch function is available only for the loader path.

6974	Maximum value of the operating range of the 17-th position switch
to	to
6981	Maximum value of the operating range of the 24-th position switch

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the maximum value of the operating range of the seventeenth to twenty-fourth position switches.

> **NOTE**
> The 17-th to 24-th position switch function is available only for the loader path.

6982	Minimum value of the operating range of the 17-th position switch
to	To
6989	Minimum value of the operating range of the 24-th position switch

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the minimum value of the operating range of the seventeenth to twenty-fourth position switches.

> **NOTE**
> The 17-th to 24-th position switch function is available only for the loader path.

	#7	#6	#5	#4	#3	#2	#1	#0
8100	NWP						IAL	RST

[Input type] Parameter input
[Data type] Bit machine group

#0 RST Pressing the reset key on the MDI unit is:
0: Valid for two paths.
1: Valid only for the path selected by the path selection signal.

The reset key on the MDI unit is valid for all machine groups. So, in machine groups for which this parameter is set to 0, a reset can be performed for all paths. In machine groups for which this parameter is set to 1, a reset can be performed only for the path that is selected by the path selection signal.

#1 IAL Chose of an option concerning operation continuation when an alarm is issued, and choice of an option concerning the start of automatic operation in alarm state:
0: When an alarm is issued, the operation is stopped with the other path(s) in same group placed in hold state.
When the other path or paths in same group are placed in alarm state, automatic operation cannot be started.
1: Even when an alarm is issued, the operation is continued without stopping the other path(s).
Even when the other path or paths in same group are placed in alarm state, automatic operation can be started.

#7 NWP Servo activation is turned on:
0: Together with other machine groups. (Servo activation is not turned on until other machine groups are ready to turn on servo activation.)
1: Independently of other machine groups. (Each machine group turns on servo activation even if other machine groups are not ready to turn on servo activation.)

	#7	#6	#5	#4	#3	#2	#1	#0
8103							MWP	MWT

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 MWT As the signal interface for the waiting M code:
0: The path individual signal interface is used.
1: The path common signal interface is used.

#1 MWP To specify a P command for the waiting M code:
0: A binary value is used as conventionally done.
1: A path number combination is used.

	#7	#6	#5	#4	#3	#2	#1	#0
8106					SCD			

[Input type] Parameter input
[Data type] Bit

#3 SCD When the function for loader control is valid, path selection by pressing MDI keys SHIFT and HELP at the same time is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
11362							LCI	

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 LCI When the function for loader control is valid, a path name at the lower right of the screen is displayed in:
0: Normal display.
1: Reverse display.

> **NOTE**
> This parameter is available only for the loader path.

Alarm and message

Number	Message	Description
PS0010	IMPROPER G-CODE	An unusable G code is specified.

Number	Message	Description
PS0365	TOO MANY MAXIMUM SV/SP AXIS NUMBER PER PATH	The maximum control axis number or maximum control spindle number which could be used within a path was exceeded.
PW0001	X-ADDRESS IS NOT ASSIGNED	The X address of the PMC could not be assigned correctly. This alarm may occur in the following case: - During the setting of parameter No. 3013, the X address could not be assigned correctly for the deceleration dog (*DEC) for a return to the reference position. When there are 4 or more paths or there are 9 or more axes for one path, the signals must be assigned by setting bit 2 (XSG) of parameter No. 3008, parameter No. 3013, and parameter No. 3014.

9 AUXILIARY FUNCTION

9.1 AUXILIARY FUNCTION/2ND AUXILIARY FUNCTION

Overview

- Auxiliary function (M code)

When up to eight digits are specified after M code, a code signal and a strobe signal are sent to the machine. The machine uses these signals to turn on or off its functions.
Usually, only one M code can be specified in one block. In some cases, however, up to five M codes can be specified for some types of machine tools (see "MULTIPLE M COMMANDS IN ONE BLOCK")
Also, parameter No. 3030 can specify the maximum digits and if a specified value exceeds the maximum digits, an alarm may be issued.

- 2nd auxiliary function (B code)

When up to eight digits are specified after address B, a code signal and strobe signal are sent. These signals are used to index the rotation axis of the machine. The code signal is retained until another B code is specified.
In each block, a single B code can be specified. The maximum number of digits that can be specified after address B is specified in parameter No. 3033. If a specified value exceeds the maximum digits, an alarm may be issued.
The address for specifying the 2nd auxiliary function can be changed from B to another address (A, C, U, V, or W) by parameter setting (parameter No. 3460).

Explanation

- Command format of 2nd auxiliary function

Command range

-99999999 to 99999999

Command method

The value specified after the address of the second auxiliary function is output on the code signals B00 to B31. Note the following about a output value.

1. When a command with a decimal point or a negative command is disabled
(When bit 0 (AUP) of parameter No. 3450 is set to 0)

   When the second auxiliary function with no decimal point is specified, the specified value is output on the code signals as is, regardless of the desktop calculator decimal point setting (bit 0 (DPI) of parameter No. 3401).
   Example:

Specified value	Output value
B10	10

   When the second auxiliary function with a decimal point is specified, alarm PS0007 is issued.
   When the second auxiliary function is specified with a negative value, alarm PS0006 is issued.

2. When a command with a decimal point or a negative command is enabled
(When bit 0 (AUP) of parameter No. 3450 is set to 1)

When the desktop calculator decimal point setting is not specified (when bit 0 (DPI) of parameter No. 3401 is set to 0), if the second auxiliary function with no decimal point is specified, the specified value is output on the code signals as is.
Example:

Specified value	Output value
B10	10

When desktop calculator decimal point input is specified (when bit 0 (DPI) of parameter No. 3401 is set to 1), if the second auxiliary function with no decimal point is specified, the specified value multiplied by a magnification is output on the code signals. (Magnifications are shown in Table 9.1 (a).)
Example:

Specified value	Output value
B10	10000 (When metric input is used and the reference axis is IS-B. The magnification is 1000.)

When the second auxiliary function with a decimal point is specified, the specified value multiplied by a magnification is output to the code signals. (Magnifications are shown in Table 9.1 (a).)
Example:

Specified value	Output value
B10.	10000 (When metric input is used and the reference axis is IS-B. The magnification is 1000.)
B0.123	1230 (When inch input is used, the reference axis is IS-B, and parameter AUX is set to 1. The magnification is 10000.)

The magnification is determined as shown below (Table 9.1 (a)) according to the setting unit of the reference axis (specified by parameter No. 1031) and bit 0 (AUX) of parameter No. 3405.

Table 9.1 (a) Magnifications for an output value when the second auxiliary function with a decimal point is specified for desktop calculator decimal point input

Setting unit		Parameter AUX = 0	Parameter AUX = 1
Metric input system	Reference axis: IS-A	100×	100×
	Reference axis: IS-B	1000×	1000×
	Reference axis: IS-C	10000×	10000×
Inch input system	Reference axis: IS-A	100×	1000×
	Reference axis: IS-B	1000×	10000×
	Reference axis: IS-C	10000×	100000×

⚠ **CAUTION**
If a decimal fraction remains after multiplying the specified value with a decimal point by a magnitude in Table 9.1 (a), the fraction is truncated.
Example:

Specified value	Output value
B0.12345	1234 (When inch input is used, the reference axis is IS-B, and parameter AUX is set to 1. The magnification is 10000.)

NOTE

If the number of digits of the specified value exceeds the allowable number of digits (set by parameter No. 3033), alarm PS0003 is issued.
When the specified value is multiplied by a magnitude in Table 9.1 (a), the allowable number of digits must be set for the resultant value.

Basic procedure

The following signals (Table 9.1 (b)) are used with these functions. (For details of the spindle-speed function and tool function, see Chapters “SPINDLE SPEED FUNCTION” and “TOOL FUNCTION”.)

Table 9.1 (b)

Function	Program address	Output signal			Input signal
		Code signal	Strobe signal	Distribution end signal	Completion signal
Auxiliary function	M	M00 to M31	MF	DEN	FIN
Spindle speed function	S	S00 to S31	SF		
Tool function	T	T00 to T31	TF		
2nd auxiliary function	B	B00 to B31	BF		

Each function uses different program addresses and different signals, but they all input and output signals in the same way, as described below. (A sample procedure for the auxiliary function is described below. The procedures for the spindle speed function, tool function, and 2nd auxiliary function, are obtained simply by substituting S, T, or B in place of M.)

<1> Suppose that Mxxx is specified in a program.
For xxx, the number of specifiable digits is specified in parameters Nos. 3030 to 3033 for each function. If a specified value exceeds the maximum digits, an alarm may be issued.

<2> Code signal M00 to M31 is sent to machine interface. After period TMF, specified in parameter No. 3010 (standard value: 16 msec), the strobe signal MF is set to 1. The code signal is the binary representation of the programmed value xxx.(*1) If a move, dwell, spindle, or other function is specified in the same block as the auxiliary function, the execution of the other function is started when the code signal of the auxiliary function is sent.

<3> When the strobe signal is set to 1, read the code signal and perform the corresponding operation on the PMC side.

<4> To execute an operation after the completion of the move, dwell or other function specified in the block, wait until distribution end signal DEN is set to 1.

<5> Upon completion of the operation, set completion signal FIN to 1 on the PMC side. The completion signal is used by the auxiliary function, spindle function, tool function, 2nd auxiliary function, external operation function described later, and other functions. If any of these functions are executed simultaneously, the completion signal must be set to 1 upon completion of all the functions.

<6> If the completion signal remains set to 1 for longer than period TFIN, specified in parameter No. 3011 (standard value: 16 msec), the CNC sets the strobe signal to 0 and reports that the completion signal has been received.

<7> When the strobe signal is set to 0, set the completion signal to 0 in the PMC.

<8> When the completion signal is set to 0, the CNC sets all code signals to 0 and completes all sequences of the auxiliary function.(*2)

<9> Once all other commands in the same block have been completed, the CNC executes the next block.

(*1) When the tool function is executed, the programmed tool number is sent as the code signal (lathe series).

(*2) When the spindle-speed function, tool function, or 2nd auxiliary function is executed, the code signal is maintained until a new code for the corresponding function is specified.

The timing diagram is shown below (Fig. 9.1):

Example 1 Single auxiliary function specified in a block

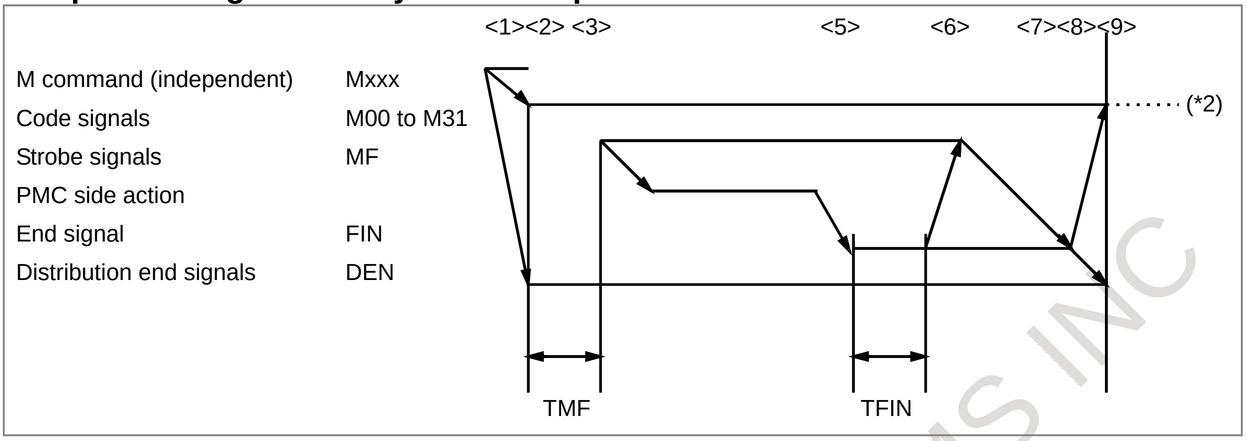
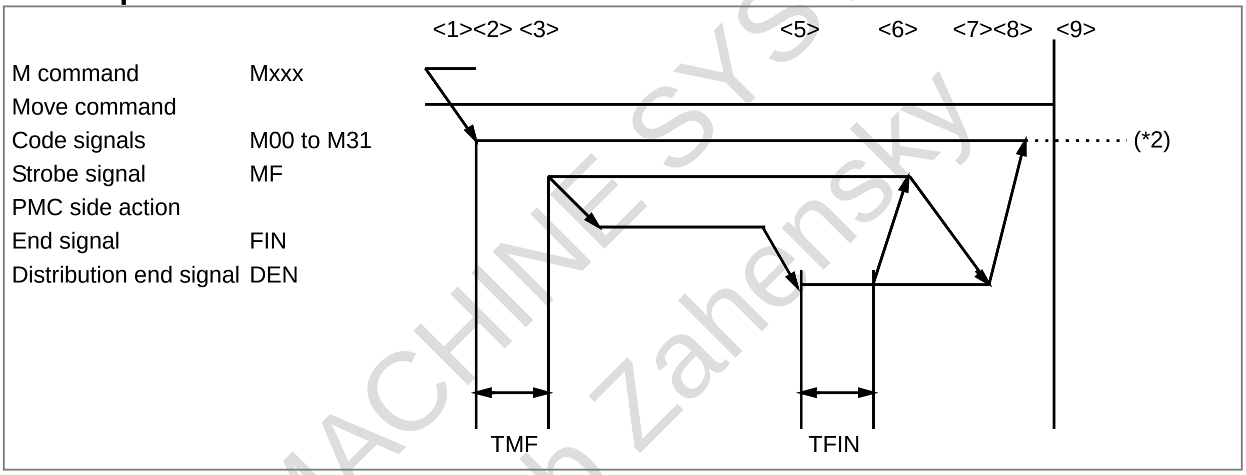


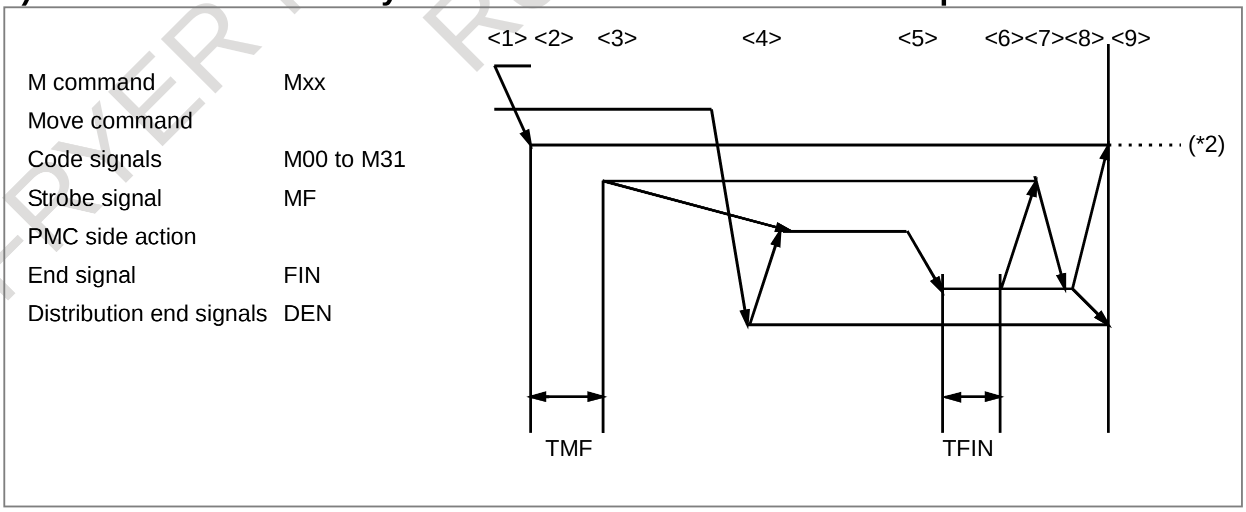
Fig. 9.1

Example 2 Move command and auxiliary function in the same block

a) Execution of a auxiliary function without waiting for move command completion



b) Execution of a auxiliary function after move command completion



Signal

End signal FIN<Gn004.3>

[Classification] Input signal

[Function] This signal reports the completion of a auxiliary function, spindle function, tool function, 2nd auxiliary function, or external operation function.

[Operation] For the control unit operation and procedure when this signal goes “1”, see the description of "Basic procedure" above.
The FIN signal must remain “1” for a certain time (TFIN, which is set by a parameter No. 3011) or longer. The FIN signal driven “1” is ignored if the FIN signal goes “0” before TFIN elapses.

> ⚠ **CAUTION**
> Only one end signal is used for all functions above. The end signal must go “1” after all functions are completed.

2nd auxiliary function end signal BFIN<Gn005.7>

[Classification] Input signal

[Function] This signal reports the completion of a 2nd auxiliary function using the high-speed M/S/T/B interface.

[Operation] For the control unit operation and procedure when this signal goes “1”, see the description of "Basic procedure" above.

Distribution end signal DEN<Fn001.3>

[Classification] Output signal

[Function] These signals report that all commands (such as move commands and dwell) are completed except those auxiliary functions, spindle functions, 2nd auxiliary functions tool functions, and so forth that are contained in the same block and have been sent to the PMC. They also report that the end signal from the PMC is being awaited.

[Output cond.] The DEN signal turns to “1” when:

- Waiting for the completion of auxiliary functions, spindle functions, tool functions, and 2nd auxiliary functions, and all other commands in the same block are completed, and the current position is in the in-position.

The DEN signal turns to “0” when:

- The execution of one block is completed

> **NOTE**
> 1 Bit 5 (NCI) of parameter No. 1601 can specify, whether to only check if an acceleration/deceleration delay is eliminated, or to also check if a servo delay (error) has been reduced to within a certain range.
> 2 When the high-speed MSTB interface is used, the distribution end signal DEN may be set to “0” when the execution of one block is completed. To do this, set bit 2 (CHM) of parameter No. 3002 to 1.

Decode M signals
DM00<Fn009.7>,DM01<Fn009.6>,DM02<Fn009.5>,DM30<Fn009.4>

[Classification] Output signal

[Function] These signals report particular auxiliary functions are specified. The auxiliary functions in a command program correspond to output signals as indicated below.

Command program	Output signal
M00	DM00
M01	DM01
M02	DM02
M30	DM30

[Output cond.] A decode M signal goes “1” when:

- The corresponding auxiliary function is specified, and any move commands and dwell commands specified in the same block are completed. These signals are not output when the end signal of the auxiliary function is returned before completion of such move commands and dwell commands.

A decode M signal goes “0” when:

- The FIN signal goes “1”
- Reset occurs

Auxiliary function code signals M00 to M31<Fn010 to Fn013>
Auxiliary function strobe signal MF<Fn007.0>

[Classification] Output signal

[Function] These signals report the specification of auxiliary functions.

[Output cond.] For the output conditions and procedure, see the description of "Basic procedure" above.

> **NOTE**
> 1 The following auxiliary functions are only processed internally by the control unit; they are not output to the PMC when programmed:
>   - M98, M99, M198
>   - M code that calls a sub program (parameters Nos. 6071 to 6079)
>   - M code that calls a custom macro (parameters Nos. 6080 to 6089)
>
> 2 Decode signals as well as the code signals and strobe signal are output for the auxiliary functions listed below.
> M00, M01, M02, M30

Spindle function code signals S00 to S31<Fn022 to Fn025>
Spindle function strobe signal SF<Fn007.2>

[Classification] Output signal

[Function] These signals report that spindle functions have been specified.

[Output cond.] For the output conditions and procedure, see the description of "Basic procedure" above.

Tool function code signals T00 to T31<Fn026 to Fn029>
Tool function strobe signal TF<Fn007.3>

[Classification] Output signal

[Function] These signals report that tool functions have been specified.

[Output cond.] For the output conditions and procedure, see the description of "Basic procedure" above.

2nd auxiliary function code signals B00 to B31<Fn030 to Fn033>
2nd auxiliary function strobe signal BF<Fn007.7>

[Classification] Output signal
[Function] These signals report that second auxiliary functions have been specified.
[Output cond.] For the output conditions and procedure, see the description of "Basic procedure" above.

Signal address

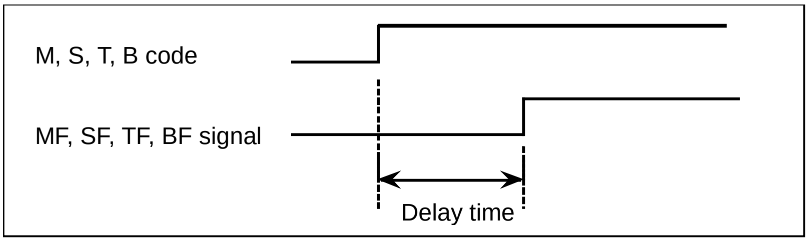
	#7	#6	#5	#4	#3	#2	#1	#0
Gn004					FIN			
Gn005	BFIN							

	#7	#6	#5	#4	#3	#2	#1	#0
Fn001					DEN			
Fn007	BF				TF	SF		MF
Fn009	DM00	DM01	DM02	DM30				
Fn010	M07	M06	M05	M04	M03	M02	M01	M00
Fn011	M15	M14	M13	M12	M11	M10	M09	M08
Fn012	M23	M22	M21	M20	M19	M18	M17	M16
Fn013	M31	M30	M29	M28	M27	M26	M25	M24
Fn022	S07	S06	S05	S04	S03	S02	S01	S00
Fn023	S15	S14	S13	S12	S11	S10	S09	S08
Fn024	S23	S22	S21	S20	S19	S18	S17	S16
Fn025	S31	S30	S29	S28	S27	S26	S25	S24
Fn026	T07	T06	T05	T04	T03	T02	T01	T00
Fn027	T15	T14	T13	T12	T11	T10	T09	T08
Fn028	T23	T22	T21	T20	T19	T18	T17	T16
Fn029	T31	T30	T29	T28	T27	T26	T25	T24
Fn030	B07	B06	B05	B04	B03	B02	B01	B00
Fn031	B15	B14	B13	B12	B11	B10	B09	B08
Fn032	B23	B22	B21	B20	B19	B18	B17	B16
Fn033	B31	B30	B29	B28	B27	B26	B25	B24

Parameter

3010	Time lag in strobe signals MF, SF, TF, and BF

[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 32767
The time required to send strobe signals MF, SF, TF, and BF after the M, S, T, and B codes are sent, respectively.

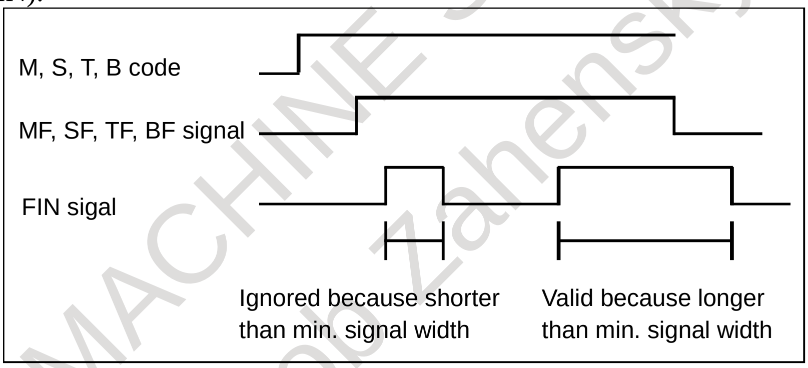


NOTE
The time is counted in units of 8ms. If the set value is not a multiple of eight, it is raised to the next multiple of eight
Example
When 30 is set, 32 ms is assumed.
When 0 is set, 8ms is assumed.

3011	**Acceptable width of M, S, T, and B function completion signal (FIN)**

[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 32767

Set the minimum signal width of the valid M, S, T, and B function completion signal (FIN).



NOTE
The time is counted in units of 8ms. If the set value is not a multiple of eight, it is raised to the next multiple of eight
Example
When 30 is set, 32 ms is assumed.
When 0 is set, 8ms is assumed.

3030	**Allowable number of digits for the M code**
3031	**Allowable number of digits for the S code**
3032	**Allowable number of digits for the T code**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 8

Set the allowable numbers of digits for the M, S, and T codes.
When 0 is set, the allowable number of digits is assumed to be 8.

3033	Allowable number of digits for the B code (second auxiliary function)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 8

Set the allowable number of digits for the second auxiliary function.
When 0 is set, the allowable number of digits is assumed to be 8.
To enable a decimal point to be specified, bit 0 (AUP) of parameter No. 3450 must be set to 1. In this case, the allowable number of digits set in this parameter includes the number of decimal places.
If a value exceeding the allowable number of digits is specified, the alarm PS0003 is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
3401								DPI

[Input type] Parameter input
[Data type] Bit path

#0 DPI When a decimal point is omitted in an address that can include a decimal point
0: The least input increment is assumed. (Normal decimal point input)
1: The unit of mm, inches, degree, or second is assumed. (Pocket calculator type decimal point input)

	#7	#6	#5	#4	#3	#2	#1	#0
3404			M02	M30				

[Input type] Parameter input
[Data type] Bit path

#4 M30 When M30 is specified in a memory operation:
0: M30 is sent to the machine, and the head of the program is automatically searched for. So, when the ready signal FIN is returned and a reset or reset and rewind operation is not performed, the program is executed, starting from the beginning.
1: M30 is sent to the machine, but the head of the program is not searched for. (The head of the program is searched for by the reset and rewind signal.)

#5 M02 When M02 is specified in memory operation:
0: M02 is sent to the machine, and the head of the program is automatically searched for. So, when the end signal FIN is returned and a reset or reset and rewind operation is not performed, the program is executed, starting from the beginning.
1: M02 is sent to the machine, but the head of the program is not searched for. (The head of the program is searched for by the reset and rewind signal.)

	#7	#6	#5	#4	#3	#2	#1	#0
3405								AUX

[Input type] Parameter input
[Data type] Bit path

#0 AUX When the second auxiliary function is specified in the calculator-type decimal point input format or with a decimal point, the multiplication factor for a value output (onto the code signal) relative to a specified value is such that:

0: The same multiplication factor is used for both of metric input and inch input.
1: A multiplication factor used for inch input is 10 times greater than that used for metric input.

When the second auxiliary function is specified in the calculator-type decimal point input format or with a decimal point, the value output onto the code signal is a specified value multiplied by a value indicated below.

Increment system		Parameter AUX=0	Parameter AUX=1
Metric input system	IS-A for reference axis	100 times	100 times
	IS-B for reference axis	1000 times	1000 times
	IS-C for reference axis	10000 times	10000 times
Inch input system	IS-A for reference axis	100 times	1000 times
	IS-B for reference axis	1000 times	10000 times
	IS-C for reference axis	10000 times	100000 times

3411	M code preventing buffering 1
3412	M code preventing buffering 2
3413	M code preventing buffering 3
to	to
3420	M code preventing buffering 10

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999

Set M codes that prevent buffering the following blocks. If processing directed by an M code must be performed by the machine without buffering the following block, specify the M code.
M00, M01, M02, and M30 always prevent buffering even when they are not specified in these parameters.

3421	Range specification 1 of M codes that do not perform buffering (lower limit)
3422	Range specification 1 of M codes that do not perform buffering (upper limit)
3423	Range specification 2 of M codes that do not perform buffering (lower limit)
3424	Range specification 2 of M codes that do not perform buffering (upper limit)
3425	Range specification 3 of M codes that do not perform buffering (lower limit)
3426	Range specification 3 of M codes that do not perform buffering (upper limit)
3427	Range specification 4 of M codes that do not perform buffering (lower limit)
3428	Range specification 4 of M codes that do not perform buffering (upper limit)
3429	Range specification 5 of M codes that do not perform buffering (lower limit)
3430	Range specification 5 of M codes that do not perform buffering (upper limit)
3431	Range specification 6 of M codes that do not perform buffering (lower limit)
3432	Range specification 6 of M codes that do not perform buffering (upper limit)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999

When a specified M code is within the range specified with parameters Nos. 3421 and 3422, 3423 and 3424, 3425 and 3426, 3427 and 3428, 3429 and 3430, or 3431 and 3432, buffering for the next block is not performed until the execution of the block is completed.

NOTE
M00, M01, M02, and M30 are M codes that do not perform buffering, regardless of parameter setting.
M98, M99, M codes for calling subprograms, and M codes for calling custom macros are M codes that performs buffering, regardless of parameter setting.

3436	Range specification 1 of second auxiliary function codes that do not perform buffering (lower limit)
3437	Range specification 1 of second auxiliary function codes that do not perform buffering (upper limit)
3438	Range specification 2 of second auxiliary function codes that do not perform buffering (lower limit)
3439	Range specification 2 of second auxiliary function codes that do not perform buffering (upper limit)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 99999999
Set the upper limit and lower limit of a series of second auxiliary function codes that do not perform buffering.
These parameters are invalid if the setting of an upper limit conflicts with the setting of a lower limit.

	#7	#6	#5	#4	#3	#2	#1	#0
3450	BDX							AUP

[Input type] Parameter input
[Data type] Bit path

#0 AUP The second auxiliary function specified in the calculator-type decimal point input format, with a decimal point, or with a negative value is:
0: Disabled.
1: Enabled.
If the second auxiliary function is specified after setting this bit to 0, the following operation results:
1. When a value is specified without a decimal point
A specified value is output onto the code signal without modification, regardless of the setting of the calculator-type decimal point input format (with bit 0 (DPI) of parameter No. 3401).
2. When a value is specified with a decimal point
The alarm PS0007, “ILLEGAL USE OF DECIMAL POINT” is issued.
3. When a negative value is specified
The alarm PS0006, “ILLEGAL USE OF MINUS SIGN” is issued.

#7 BDX When ASCII code is called using the same address as the address for the second auxiliary function (specified by parameter No. 3460), this parameter prevents the difference between the argument unit used when the second auxiliary function is enabled (bit 2 (BCD) of parameter No.8132 is 1) and the unit when the function is disabled (bit 2 (BCD) of parameter Np.8132 is 0).

0: When bit 0 (AUP) of parameter No. 3450 is set to 1, the argument unit differs, depending on whether the second auxiliary function is enabled or disabled.
1: The same argument unit is used. (The unit applied when the second auxiliary function is enabled is used.)

3460	Second auxiliary function specification address

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 65to67, 85to87

Specify which of A, B, C, U, V, and W is to be used as the address for specifying the second auxiliary function. If an address used as an axis name is specified, the second auxiliary function is disabled.

Name	A	B	C	U	V	W
Setting value	65	66	67	85	86	87

Address B is assumed when a value other than the above is set.
However, the name U, V, or W can be used with the T series only when G code system B or C is used. When a value from 85 to 87 is specified with G code system A, the specification address for the second auxiliary function is B.

11290	M code preventing buffering 11
11291	M code preventing buffering 12
11292	M code preventing buffering 13
to	to
11299	M code preventing buffering 20

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999999

Set M codes that prevent buffering the following blocks. If processing directed by an M code must be performed by the machine without buffering the following block, specify the M code.
M00, M01, M02, and M30 always prevent buffering even when they are not specified in these parameters.

Note

NOTE

1 When a move command and auxiliary function are specified in the same block, the commands are executed in one of the following two ways:
(1) Simultaneous execution of the move command and auxiliary function commands.
(2) Executing auxiliary function commands upon completion of move command execution.
The selection of either sequence depends on the sequence of PMC.

2 When the 2nd auxiliary function is provided, the address used for specifying the 2nd auxiliary function (B or the address specified with parameter No. 3460) cannot be used as an axis address.

3 The block following M00, M01, M02 and M30, is not read into the input buffer register, if present. Similarly, ten M codes which do not buffer can be set by parameters Nos. 3411 to 3420 and 11290 to 11299.

NOTE

4 For M00 and M01, auxiliary function code signal, auxiliary function strobe signal, and M decode signals are only sent; the control of program stop and optional stop shall be designed on the PMC side.

5 When the automatic operation is stopped by M02 or M30, it is necessary to send the external reset signal from the machine side to the CNC, instead of the FIN signal. When the external reset signal is returned against the M02 or M30, the control returns to the top of the program recently executed, and enters the reset state. When the FIN signal is returned, the control returns to the beginning of the program recently executed, and executes it from the top.

6 In programmable data entry (G10)/programmable parameter entry (G10/G11) command blocks and modes, auxiliary function code (M, S, T, and B) commands are disabled. If a specified code is not in the G10 command format, alarm PS1144, "G10 FORMAT ERROR", is issued.

Alarm and message

Number	Message	Description
PS0003	TOO MANY DIGIT	Data entered with more digits than permitted in the NC instruction word. The number of permissible digits varies according to the function and the word.
PS0006	ILLEGAL USE OF MINUS SIGN	A minus sign (–) was specified at an NC instruction word or system variable where no minus signal may be specified.
PS0007	ILLEGAL USE OF DECIMAL POINT	A decimal point (.) was specified at an address where no decimal point may be specified, or two decimal points were specified.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Auxiliary function (M code)
	2nd auxiliary function (B code)

9.2 AUXILIARY FUNCTION LOCK

Overview

Inhibits execution of a specified M, S, T and B function. That is, code signals and strobe signals are not issued. This function is used to check a program with machine lock.

Signal

Auxiliary function lock signal AFL<Gn005.6>

[Classification] Input signal

[Function] This signal selects auxiliary function lock. That is, this signal disables the execution of specified M, S, T, and B functions.

[Operation] When this signal turns to "1", the control unit functions as described below.

(1) The control unit does not execute M, S, T, and B functions specified for memory operation, DNC operation, or MDI operation. That is, the control unit stops the output of code signals and strobe signals (MF, SF, TF, BF).

(2) If this signal turns to "1" after code signal output, the output operation is executed in the ordinary manner until its completion (that is, until the FIN signal is received, and the strobe signal turns to "0").

(3) Among the auxiliary functions, M00, M01, M02, and M30 are executed even when this signal is “1”. All code signals, strobe signals, decode signals are output in the ordinary manner.
(4) Among the auxiliary functions, even when this signal is “1”, those functions (M98 and M99) that are executed in the control unit without outputting their execution results are executed in the ordinary manner.

> ⚠ **CAUTION**
> Even when this signal is “1”, spindle analog output or spindle serial output is executed.

Auxiliary function lock check signal MAFL<Fn004.4>

[Classification] Output signal
[Function] This signal reports the state of the auxiliary function lock signal AFL.
[Output cond.] This signal turns to “1” when:
- The auxiliary function lock signal AFL is “1”

This signal turns to 0 when:
- The auxiliary function lock signal AFL is “0”

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn005		AFL						

	#7	#6	#5	#4	#3	#2	#1	#0
Fn004				MAFL				

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Machine lock and auxiliary function lock

9.3 MULTIPLE M COMMANDS IN A SINGLE BLOCK

Overview

So far, one block has been able to contain only one M code. However, this function allows up to three M codes to be contained in one block, when bit 7(M3B) of parameter No. 3404 is set to 1. And, Up to five M codes can be specified in a block, when bit 5(M5B) of parameter No. 11630 is set to 1.
Up to five M codes specified in a block are simultaneously output to the machine (PMC). This means that compared with the conventional method of a single M command in a single block, a shorter cycle time can be realized in machining.

(Example)

One M command in a single block	Multiple M commands in a single block
M40; M50; M60; M70; M80; G28G91X0Y0Z0; :	M40M50M60 M70M80; G28G91X0Y0Z0; : : : : :

Explanation

Basic procedure

(1) Assume that "MaaMbbMccMddMee;" was commanded by the program.

(2) The 1st M command (Maa) sends the code signals M00 to M31 in a manner similar to the conventional one-block single command. The strobe signal MF is set to “1” after a time TMF set by parameter No. 3010 (Standard setting: 16 msec).
The second M command (Mbb) sends the code signal M200-M231, the third M command (Mcc) sends the code signal M300-M331, the forth M command (Mdd) sends the code signal M400-M431, the fifth M command (Mee) sends the code signal M500-M531, and their respective strobe signals MF2, MF3, MF4 and MF5 are set to “1”.
Furthermore, the three code signals are sent simultaneously.
The strobe signals MF, MF2, MF3, MF4 and MF5 become “1” at the same time.
The code signal is a binary notation of the program command aa, bb and cc.

(3) On the PMC side, read the code signals corresponding to the respective strobe signals when the strobe signals become “1”, and perform the appropriate operations.

(4) When the operation of all M commands ends on the PMC side, the end signal (FIN) is set to “1”.

(5) When the completion signal stays “1” for a time (TFIN) set by parameter No. 3011 (Standard: 16 msec), all strobe signals (MF, MF2, MF3, MF4 and MF5) are set to “0” at the same time and the reception of completion signal is reported.

(6) On the PMC side, when MF, MF2, MF3, MF4 and MF5 are set to “0”, the completion signal is set to “0”.

A time chart for this procedure is shown below (Fig. 9.3):

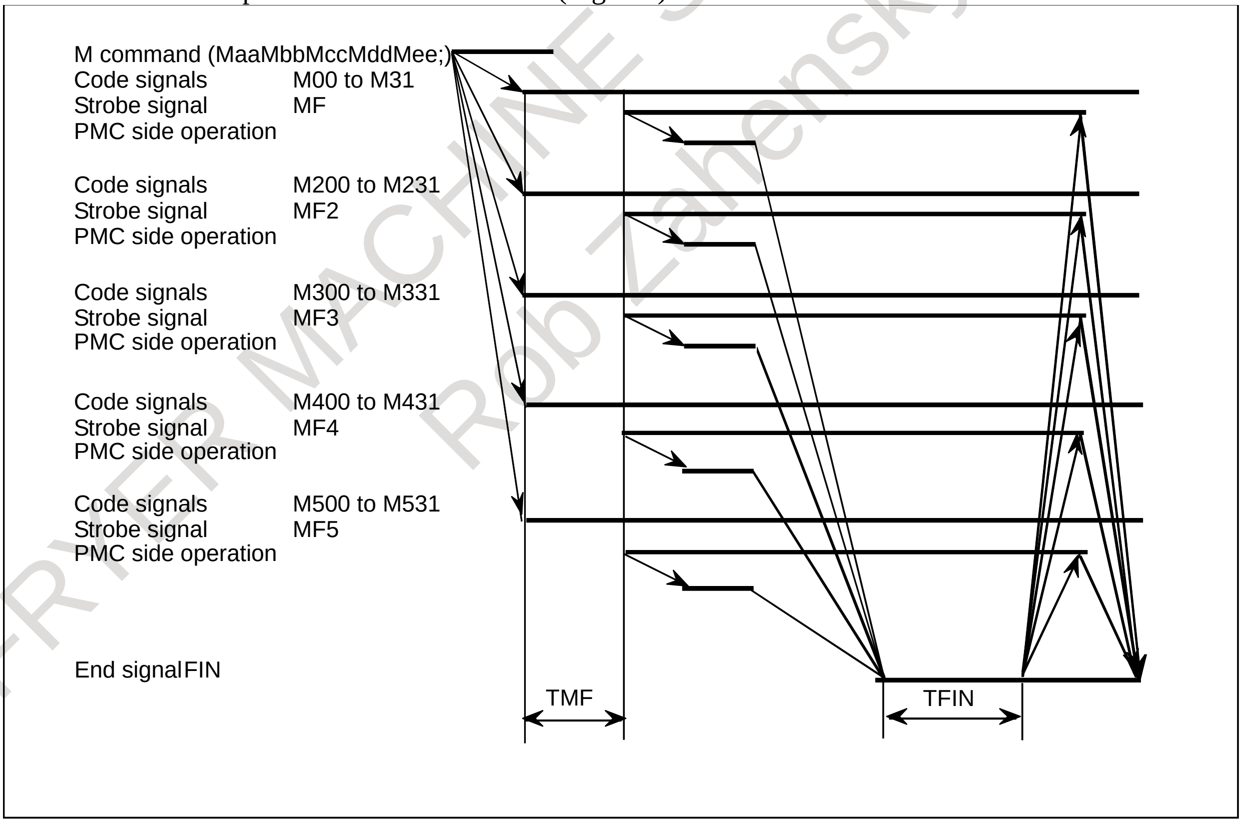


Fig. 9.3

M code preventing buffering

Usually in multiple M commands in a single block, the second M command to the fifth M command cannot be set as M code preventing buffering. By setting bit 0 (MMB) of parameter No. 11658 to 1, the second M command to the fifth M command can be set as M code preventing buffering by any of the following parameters.

- Parameters Nos. 3411 to 3420
- Parameters Nos. 3421 to 3432
- Parameters Nos. 11290 to 11299

Signal

2nd, 3rd, 4th, 5th M function code signals

When three M code in a single block is effective（bit 7(M3B) of parameter No. 3404 is set to 1）

M200 to M215<Fn014 to Fn015>,M300 to M315<Fn016 to Fn017>

When five M code in a single block is effective（bit 5(M5B) of parameter No. 11630 is set to 1）

M200 to M231<Fn014 to Fn017>, M300 to M331<Fn564 to Fn567>

M400 to M431<Fn568 to Fn571>, M500 to M531<Fn572 to Fn575>

2nd, 3rd, 4th, 5th M Function strobe signals

MF2 <Fn008.4>,MF3<Fn008.5>,MF4<Fn008.6>,MF5<Fn008.7>

[Classification] Output signal

[Function] Indicates that second, third, forth and fifth auxiliary functions have been issued.

[Output cond.] For the output conditions and procedure, see the description of “Basic Procedure” above.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn008	MF5	MF4	MF3	MF2				
Fn014	M207	M206	M205	M204	M203	M202	M201	M200
Fn015	M215	M214	M213	M212	M211	M210	M209	M208

When three M code in a single block is effective（bit 7(M3B) of parameter No. 3404 is set to 1）

	#7	#6	#5	#4	#3	#2	#1	#0
Fn016	M307	M306	M305	M304	M303	M302	M301	M300
Fn017	M315	M314	M313	M312	M311	M310	M309	M308

When five M code in a single block is effective（bit 5(M5B) of parameter No. 11630 is set to 1）

	#7	#6	#5	#4	#3	#2	#1	#0
Fn016	M223	M222	M221	M220	M219	M218	M217	M216
Fn017	M231	M230	M229	M228	M227	M226	M225	M224
Fn564	M307	M306	M305	M304	M303	M302	M301	M300
Fn565	M315	M314	M313	M312	M311	M310	M309	M308
Fn566	M323	M322	M321	M320	M319	M318	M317	M316
Fn567	M331	M330	M329	M328	M327	M326	M325	M324

Fn568	M407	M406	M405	M404	M403	M402	M401	M400
Fn569	M415	M414	M413	M412	M411	M410	M409	M408
Fn570	M423	M422	M421	M420	M419	M418	M417	M416
Fn571	M431	M430	M429	M428	M427	M426	M425	M424
Fn572	M507	M506	M505	M504	M503	M502	M501	M500
Fn573	M515	M514	M513	M512	M511	M510	M509	M508
Fn574	M523	M522	M521	M520	M519	M518	M517	M516
Fn575	M531	M530	M529	M528	M527	M526	M525	M524

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3404	M3B							

[Input type] Parameter input
[Data type] Bit path

#7 M3B The number of M codes that can be specified in one block
0: One
1: Up to three

	#7	#6	#5	#4	#3	#2	#1	#0
11630			M5B					

[Input type] Parameter input
[Data type] Bit path

#5 M5B The number of M codes that can be specified in one block
0: One (up to three when bit 7(M3B) of parameter No. 3404 is set to 1)
1: Up to five

	#7	#6	#5	#4	#3	#2	#1	#0
11658								MMB

[Input type] Parameter input
[Data type] Bit path

#0 MMB In multiple M commands in a single block, if any of the second M command to the fifth M command is specified by any of parameters Nos. 3411 to 3420, parameters Nos. 3421 to 3432 or parameters Nos. 11290 to 11299, the M command is:
0: Not recognized as M code preventing buffering
1: Recognized as M code preventing buffering

⚠ **CAUTION**
1 Regardless of setting of bit 0 (MMB) of parameter No. 11658, the first command in multiple commands in a single block can be set as M code preventing buffering.

> ⚠ **CAUTION**
> 2 Though bit 0 (MMB) of parameter No. 11658 is set to 1, M00, M01, M02, M30 or M code preventing buffering specified by any of parameters other than the above must be specified alone in a single block.
> If any of these M codes is specified to any of the second command to the fifth command in multiple M commands in a single block, the M command is not recognized as M code preventing buffering.

Caution

> ⚠ **CAUTION**
> 1 M00, M01, M02, M30, M98, M99, or M198 must not be specified together with another M code.
> 2 Some M codes other than M00, M01, M02, M30, M98, M99, and M198 cannot be specified together with other M codes; each of those M codes must be specified alone in a single block. Such M codes include those that make the CNC perform internal operations in addition to sending the M code signals to the PMC. For example, such M codes are M codes for calling program numbers 9001 to 9009 and M codes for disabling advance reading (buffering) of subsequent blocks.
> The M codes that can be specified in a single block must be those that the CNC only sends the M code signals to the PMC side.
> If bit 0 (MMB) of parameter No. 11658 is set to 1, M code preventing buffering specified by any of parameters Nos. 3411 to 3420, parameters Nos. 3421 to 3432 or parameters Nos. 11290 to 11299 can be specified together with another M code.

Note

> **NOTE**
> 1 CNC allows up to five M codes to be specified in one block. However, some M codes cannot be specified at the same time due to mechanical operation restrictions. For example, M42 can be specified only after the mechanical operation of M41 is completed.
> 2 When bit 7(M3B) of parameter No. 3404 is set to 1, the 1st M code can be up to 8 digits and 2nd, 3rd M codes can be the values up to 65535. When bit 5(M5B) of parameter No. 11630 is set to 1, the 1st to 5th M code can be up to 8 digits.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Multiple M commands in a single block

9.4 HIGH-SPEED M/S/T/B INTERFACE

Overview

To accelerate M/S/T/B function execution, the high-speed M/S/T/B interface has simplified the transfer of the strobe and completion signals of the M/S/T/B functions.
By shortening the execution time of M / S / T / B function, machining time can be reduced.
Whether to use the conventional system or high-speed system for strobe signal and completion signal handling can be specified by bit 7 (MHI) of parameter No. 3001.

The description below uses the auxiliary functions (M code commands) as an example. The same description applies to the spindle function (S code), tool function (T code) and 2nd auxiliary function (B code).

Explanation

- Basic procedure

(1) Assume that the following program is given:
Mxx;
Myy;

(2) In response to an M command, the CNC system sends out the code signals M00 to M31.
The CNC system inverts the logical level of the strobe signal MF, that is, from “0” to “1”, or from “1” to “0”.

(3) The CNC system inverts the strobe signal, then when the logical level of the auxiliary function completion signal MFIN becomes the same as the strobe signal, the CNC assumes the completion of PMC sequence.
With the usual method, the operation is assumed to be completed when a falling edge (“1” to “0”) of the M/S/T/B completion signal FIN is received after a rising edge (“0” to “1”) of the FIN signal is detected. This new system, on the other hand, assumes the operation has been completed upon detection of only one transition of the completion signal.
In addition, the conventional system uses only one completion signal (FIN) common to the M/S/T/B functions. This new system uses a different completion signal for each of the M, S, T, and B functions; the completion signals for the M, S, T, and B functions are MFIN, SFIN, TFIN, and BFIN, respectively.

The Fig. 9.4 (a) below shows the timing chart of these signals with the new system. For comparison, Fig. 9.4 (b) shows the timing chart of the conventional system.

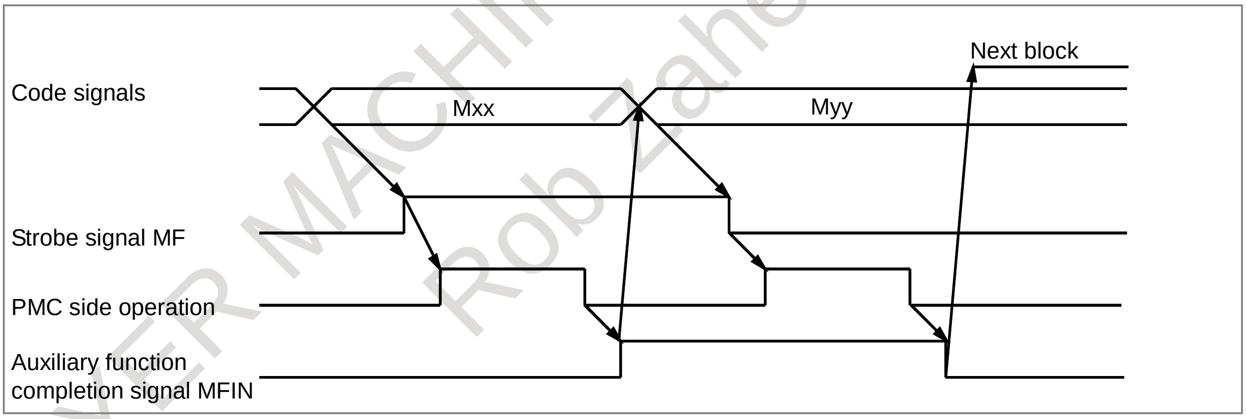


Fig. 9.4 (a) Timing chart of the high-speed system

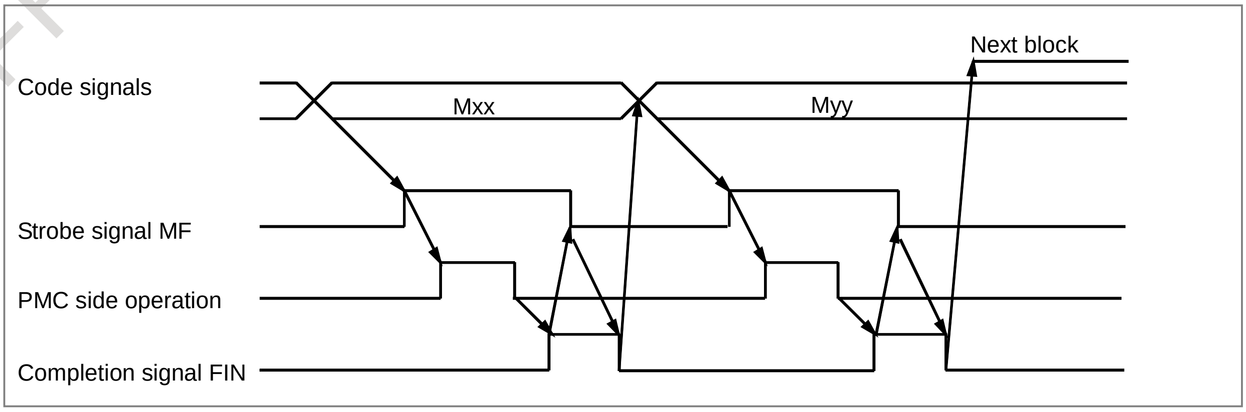


Fig. 9.4 (b) Timing chart of the conventional system

A high-speed interface can also be used for multiple M commands issued for one block. This interface provides separate completion signals for each of the 5 M codes. They are called MFIN (the same name as for the single M command per block function), MFIN2, MFIN3, MFIN4 and MFIN5 respectively. The signal transfer sequence for multiple M codes per block is the same as that for a single M code per block.
The high-speed interface can also be used for the external operation function. In this case, special external operation signal EFD and completion signal EFIN are used.
The procedure for sending and receiving these signals is identical to that for sending and receiving the strobe and completion signals of the auxiliary function (machining center system).

Signal

Auxiliary function completion signal MFIN<Gn005.0>

[Classification] Input signal
[Function] Reports that the execution of a auxiliary function using the high-speed M/S/T/B interface is completed.
[Operation] For the operation and procedure of the control unit when this signal turns to “1” or “0”, see the description of "Basic procedure" above.

Spindle function completion signal SFIN<Gn005.2>

[Classification] Input signal
[Function] Reports that the execution of a spindle speed function using the high-speed M/S/T/B interface is completed.
[Operation] For the operation and procedure of the control unit when this signal turns to “1” or “0”, see the description of "Basic procedure" above.

Tool function completion signal TFIN<Gn005.3>

[Classification] Input signal
[Function] Reports that the execution of a tool function using the high-speed M/S/T/B interface is completed.
[Operation] For the operation and procedure of the control unit when this signal turns to “1” or “0”, see the description of "Basic procedure" above.

2nd auxiliary function completion signal BFIN<Gn005.7>

[Classification] Input signal
[Function] Reports that the execution of a second auxiliary function using the high-speed M/S/T/B interface is completed.
[Operation] For the operation and procedure of the control unit when this signal turns to “1” or “0”, see the description of "Basic procedure" above.

2nd, 3rd, 4th and 5th M function completion signals MFIN2<Gn004.4>,MFIN3<Gn004.5>, MFIN4<Gn004.6>, MFIN5<Gn004.7>

[Classification] Input signal
[Function] Indicate that when the high-speed M/S/T/B interface is used for multiple M commands per block, the 2nd to 5th M functions have been completed.
[Operation] For the operation and procedure of the control unit when this signal turns to “1” or “0”, see the description of "Basic procedure" above.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn004	MFIN5	MFIN4	MFIN3	MFIN2				
Gn005	BFIN				TFIN	SFIN		MFIN

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3001	MHI							

[Input type] Parameter input
[Data type] Bit path

#7 MHI Exchange of strobe and completion signals for the M, S, T, and B
0: Normal
1: High-speed

	#7	#6	#5	#4	#3	#2	#1	#0
3002								CHM

[Input type] Parameter input
[Data type] Bit path

#0 CHM For high-speed M/S/T/B, the distribution end signal DEN and an auxiliary function code signal M00 to M31 are:
0: Not turned off even upon completion of the execution of the auxiliary function.
1: Turned off upon completion of the execution of the auxiliary function.

Note

NOTE
1 The strobe signals MF, SF, TF, and BF are "0" when the power is turned on.
2 When the control unit is reset, MF, SF, TF, and BF are set to "0".
3 Using bit 0 (CHM) of parameter No. 3002, select whether to turn the distribution end signal DEN and the auxiliary function code signals M00 to M31 to "0" at the completion of one block.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	Auxiliary function, 2nd auxiliary function
	Multiple M commands in a single block
	External operation function

9.5 M CODE GROUPING FUNCTION

Overview

Classifying up to 500 M codes into up to 127 groups allows the user:
- To receive an alarm if an M code that must be specified alone is included when multiple M codes are specified in a block.
- To receive an alarm if M codes belonging to the same group are specified in the same block when multiple M codes are specified in a block.

Setting an M code group number using the setting screen

- Procedure for displaying the M code group setting screen

You can use the "M code group setting screen" to set a group number for each M code.

Display the "M code group setting screen" using the following procedure:

1 Press function key SYSTEM and the continuous menu key several times. Soft key [M CODE GROUP] appears.
2 Press soft key [M CODE GROUP].

In the “NO.” field, M codes for which an M code group can be set are displayed.
An M code group can be set for the following M codes: M00 to M99, and any 400 M codes selected from M100 and subsequent M codes. For details of how to add the 100th and subsequent M codes, see the explanation of parameters Nos. 3441 to 3444.

In the “DATA” field, the M code group number corresponding to each M code is displayed.

- Setting a group number

To set an M code group number on the “M code group setting screen,” use the following procedure:
1 Select the MDI mode.
2 Set “PARAMETER WRITING” on the setting screen to 1.
3 Display the “M code group setting screen.
4 Move the cursor to the M code to be set using page keys and cursor keys. You can also enter the number of the M code to be set and press soft key [NO.SRH] to move the cursor to the M code.
5 Enter a group number and press soft key [INPUT] or the INPUT key.

The valid range of M code group numbers is from 1 to 127 (127 groups). If a value of 0 is input, it is not registered as an M code group.

- Examples of setting parameters Nos. 3441 to 3444

In the following examples, the number of digits of an M code is 4.
<1> to <4> indicate parameters Nos. 3441 to 3444.

(1) When <1> = 300, <2> = 400, <3> = 500, and <4> = 900 are set

Number
0000 : 0099 } 100 codes
0300 : 0399 } 100 codes
0400 : 0499 } 100 codes
0500 : 0599 } 100 codes
0900 : 0999 } 100 codes

} M code groups can be set for M0000 to M0099, M0300 to M0599, and M0900 to M0999.
M codes M0300 to M0599 and M0900 to M0999 are added to the M code group setting screen.

(2) When <1> = 200, <2> = 0, <3> = 550, and <4> = 800 are set

Number
0000
:
0099
0200
:
0299
0550
:
0649
0800
:
0899

M code groups can be set for M0000 to M0099, M0200 to M0299, M0550 to M0649, and M0800 to M0899.
(The setting of parameter <2> is invalid because it is 0.)
In this case, M codes M0200 to M0299, M0550 to M0649, and M0800 to M0899 are added to the M code group setting screen.

M code group check function

When multiple M commands in a single block (enabled when bit 7 (M3B) of parameter No. 3404 is set to 1) are used, you can check the following items. You can also select whether to check the items using bit 1 (MGC) of parameter No. 3400.

(1) M code to be specified in a single block containing no other M codes
If an M code which must be specified in a single block containing no other M codes is specified together with another M code, an alarm PS5016, ”ILLEGAL COMBINATION OF M CODES” is issued.

(2) M codes in the same group
If multiple M codes in the same group are specified together, an alarm PS5016 is issued.

The valid range of group numbers is from 0 to 127 (128 groups).
Group numbers 0 and 1 have special meaning. Note the following points:

- Each M code with group number 1 is assumed to be an M code to be specified in a single block containing no other M codes.
- For each M code with group number 0, the “same group M code check” is ignored. That is, when multiple M codes with group number 0 are specified in a single block, the alarm is not issued.
- For each M code with group number 0, the “check for an M code to be specified in a single block containing no other M codes” is not ignored. That is, if an M code with group number 1 and an M code with group number 0 are specified in a single block, the alarm is issued.
- For M codes that are not to be output to the machine such as M98, M99, M198, and M codes for subprogram and macro calls (set in parameters Nos. 6071 to 6079 and Nos. 6080 to 6089 and with the macro executor), be sure to set 0 as the group number.
- For M00, M01, M02, M30, and M codes for which buffering is suppressed (set in parameters Nos. 3411 to 3432 and 11290 to 11299), be sure to set 1 as the group number.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3400							MGC	

[Input type] Parameter input
[Data type] Bit path

#1 MGC When a single block specifies multiple M commands, an M code group check is:
0: Made.
1: Not made.

3441	Start number of M codes for which an M code group can be set (1)
3442	Start number of M codes for which an M code group can be set (2)
3443	Start number of M codes for which an M code group can be set (3)
3444	Start number of M codes for which an M code group can be set (4)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0, 100to99999999

Code numbers 0 to 99 on the M code group setting screen correspond to M00 to M99. When adding M codes after the first 100 M codes, specify a start M code number in these parameters. Thus, up to 400 M codes can be added to the M code group setting screen in groups of 100 M codes starting with the set value. When 0 is set, no M codes are added to the M code group setting screen.

When setting these parameters, follow the setting condition described below. If the condition is not satisfied, no M codes are added to the M code group setting screen as in the case where 0 is set.

(Setting condition)

The settings of parameters (1) to (4) (excluding the setting of 0) must satisfy:

99 < (1), (1)+99 < (2), (2)+99 < (3), (3) +99 < (4)

Alarm and message

Number	Message	Description
PS5016	ILLEGAL COMBINATION OF M CODES	M codes which belonged to the same group were specified in a block. Alternatively, an M code which must be specified without other M codes in the block was specified in a block with other M codes.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	M Code Grouping Function

9.6 AUXILIARY FUNCTION OUTPUT IN MOVING AXIS

Overview

By specifying absolute coordinate values and auxiliary functions (M, B) at G50.9 block, the auxiliary functions are output to PMC when absolute coordinate enters the specified area in the next movement block.
G50.9 can be specified up to 2 blocks consecutively. In other words, auxiliary function output points in a movement block can be specified up to two.
Code signals and strobe signals are output to the same signal address as a usual auxiliary function.

Format

G50.9 IP_ Mm11 Mm12 Mm13 Bb11 ;

G50.9 IP_ Mm21 Mm22 Mm23 Bb21 ;

(Movement command block) Mm01 Mm02 Mm03 Bb01 Ss01 Tt01 ;

IP_ : Auxiliary function output start point. (Absolute coordinate value)

M m11 : 1st M auxiliary function (1st block)
M m12 : 2nd M auxiliary function (1st block)
M m13 : 3rd M auxiliary function (1st block)
B b11 : 2nd auxiliary function (1st block)

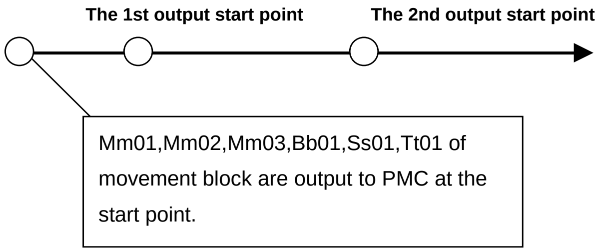
M m21 : 1st M auxiliary function (2nd block)
M m22 : 2nd M auxiliary function (2nd block)
M m23 : 3rd M auxiliary function (2nd block)
B b21 : 2nd auxiliary function (2nd block)

M m01 : 1st M auxiliary function (movement block)
M m02 : 2nd M auxiliary function (movement block)
M m03 : 3rd M auxiliary function (movement block)
B b01 : 2nd auxiliary function (movement block)
S s01 : Spindle speed command
T t01 : T command (Tool compensation)

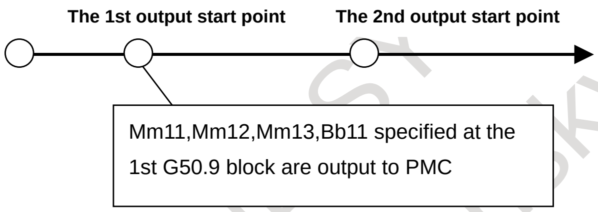
- At least one axis of auxiliary function output start point must be commanded at G50.9 block. The alarm PS5330, " G50.9 FORMAT ERROR" occurs if there is not an axis address at G50.9 block.
- At least one auxiliary function must be commanded at G50.9 block. The alarm PS5330," G50.9 FORMAT ERROR" occurs if an auxiliary function isn't commanded.
- It is possible to command G50.9 block up to 2 blocks consecutively. The alarm PS5330," G50.9 FORMAT ERROR" occurs if more than 3 blocks are commanded.
- Incremental programming cannot be specified in G50.9 block. The alarm PS0009, "IMPROPER NC ADDRESS" occurs. Also, in case of incremental mode (G91) in G code system B/C, incremental programming in G50.9 block is regarded as absolute programming.

Operation

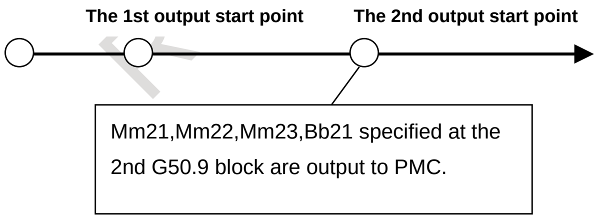
(1) At the start point of movement block, M/S/T/B codes (m01,m02,m03,b01,s01,t01) of movement block are output to PMC, and moving is started.



(2) When moving to the 1st output start point that is specified at the 1st G50.9 block, if FIN processing of
M/S/T/B code which was output to PMC in (1) has completed, M code (m11,m12,m13) and B code (b11) of the 1st G50.9 block are output to PMC.
If FIN processing has not completed, M code (m11,m12,m13) and B code (b11) are not output to PMC at the 1st output start point. They are output immediately after FIN processing completes. Code signals and strobe signals are output to the same signal address as usual auxiliary function.



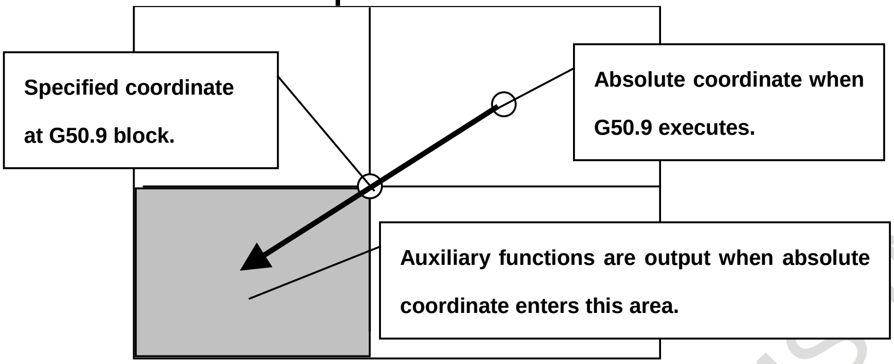
(3) When moving to the 2nd output start point that is specified at the 2nd G50.9 block, if FIN processing of M/S/T/B code which was output to PMC in (2) has completed, M code (m21,m22,m23) and B code (b21) of the 2nd G50.9 block are output to PMC.
If FIN processing has not completed, M code (m21,m22,m23) and B code (b21) are not output to PMC at the 2nd output point. They are output immediately after FIN processing completes.



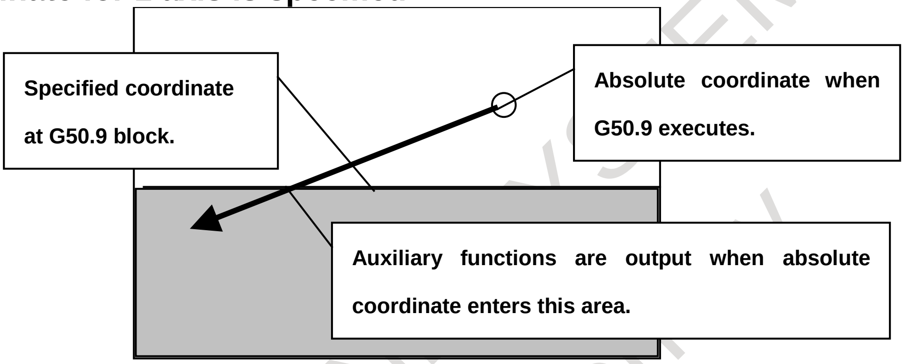
(4) When movement completes, if absolute coordinate has not reached the value that was specified at G50.9 blocks, the alarm PS5331, “ILLEGAL COMMANDED POSITION“ occurs.

Specification of output start point

When coordinates for 2 axes are specified



When coordinate for 1 axis is specified



Priority of auxiliary function output

If absolute coordinate have entered the area that was specified at the 2nd G50.9 block while they have not entered the area that was specified at the 1st G50.9 block, auxiliary function of the 2nd G50.9 block are not output at that time. After absolute coordinate entered the area that was specified at the 1st G50.9 block and FIN processing completed, auxiliary function of the 2nd G50.9 block are output finally.

Alarm at movement completion

When movement completes, if absolute coordinate has not reached the value that was specified at G50.9 blocks, the alarm PS5331, ”ILLEGAL COMMANDED POSITION” occurs.

The alarm PS5331, ”ILLEGAL COMMANDED POSITION” occurs if next block after G50.9 block is no movement command because absolute coordinate does not reach to the value that was specified at G50.9 block.

Example 1) M code

```
G00 X100. ;
G50.9 X200. M128 ;
M01;    ← The alarm PS5331 occurs.
G00 X300. ;
```

Example 2) Empty block

```
G00 X100. ;
G50.9 X200. M128 ;
; (Empty block)        ← The alarm PS5331 occurs.
G00 X300. ;
```

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3001	MHI							

[Input type] Parameter input
[Data type] Bit path

#7 MHI Handshake of strobe and completion signals for the M, S, T, and B
0: Normal
1: High-speed

	#7	#6	#5	#4	#3	#2	#1	#0
3104	DAC	DAL						

[Input type] Parameter input
[Data type] Bit path

#6 DAL The displayed absolute position is
0: The actual position that takes tool length offset into account.
1: The programmed position that does not take tool length offset into account.

> **NOTE**
> In lathe systems, whether a tool offset is excluded from displaying absolute position is determined by the setting of bit 1 (DAP) of parameter No.3129.

#7 DAC When an absolute position are displayed:
0: Values not excluding the amount of travel based on cutter compensation and tool nose radius compensation are displayed.
1: Values excluding the amount of travel based on cutter compensation and tool nose radius compensation (programmed positions) are displayed.

> **NOTE**
> When the parameter DAC is set to 1, in the command like circle interpolation where the cutter compensation vector is changed constantly, an absolute position is not correctly displayed during its interpolation except its start point and its end point.

	#7	#6	#5	#4	#3	#2	#1	#0
3129							DAP	

[Input type] Parameter input
[Data type] Bit path

#1 DAP For absolute coordinate display:
0: The actual position including a tool offset (tool movement) is displayed.
1: The programmed position excluding a tool offset (tool movement) is displayed.

> **NOTE**
> In machining center systems, whether the tool length offset is excluded from displaying absolute position is determined according to the setting of bit 6 (DAL) of parameter No.3104.

Signal

Auxiliary function code signals M00 to M31<Fn010 to Fn013>
2nd M function code signals M200 to M215<Fn014 to Fn015>
3rd M function code signals M300 to M315<Fn016 to Fn017>
Auxiliary function strobe signal MF<Fn007.0>
2nd M Function strobe signal MF2<Fn008.4>
3rd M Function strobe signal<Fn008.5>

[Classification] Output signal

[Function] These signals report that M codes have been specified.

[Output cond.] About output condition, procedure etc, please refer to "FANUC Series 0*i*-MODEL F Plus CONNECTION MANUAL (FUNCTION) 10.1 AUXILIARY FUNCTION/2ND AUXILIARY FUNCTION".

2nd auxiliary function code signals B00 to B31<Fn030 to Fn033>
2nd auxiliary function strobe signal BF<Fn007.7>

[Classification] Output signal

[Function] These signals report that second auxiliary functions have been specified.

[Output cond.] About output condition, procedure etc, please refer to "FANUC Series 0*i*-MODEL F Plus CONNECTION MANUAL (FUNCTION) 10.1 AUXILIARY FUNCTION/2ND AUXILIARY FUNCTION".

End signal FIN<Gn004.3>

[Classification] Input signal

[Function] This signal reports the completion of an auxiliary function, a spindle function, a tool function, a 2nd auxiliary function, or an external operation function.

[Operation] About input condition, procedure etc, please refer to "FANUC Series 0*i*-MODEL F Plus CONNECTION MANUAL (FUNCTION) 10.1 AUXILIARY FUNCTION/2ND AUXILIARY FUNCTION".

Auxiliary function completion signal MFIN<Gn005.0>

[Classification] Input signal

[Function] Reports that the execution of an auxiliary function using the high-speed M/S/T/B interface is completed.

[Operation] About input condition, procedure etc, please refer to "FANUC Series 0*i*-MODEL F Plus CONNECTION MANUAL (FUNCTION) 10.1 AUXILIARY FUNCTION/2ND AUXILIARY FUNCTION".

2nd M function completion signal MFIN2<Gn004.4>
3rd M function completion signal MFIN3<Gn004.5>

[Classification] Input signal

[Function] Indicate that when the high-speed M/S/T/B interface is used for multiple M commands per block, the 2nd to 3rd M functions have been completed.

[Operation] About input condition, procedure etc, please refer to "FANUC Series 0*i*-MODEL F Plus CONNECTION MANUAL (FUNCTION) 10.1 AUXILIARY FUNCTION/2ND AUXILIARY FUNCTION".

2nd auxiliary function completion signal BFIN<Gn005.7>

[Classification] Input signal

[Function] Reports that the execution of a second auxiliary function using the high-speed M/S/T/B interface is completed

[Operation] About input condition, procedure etc, please refer to "FANUC Series 0*i*-MODEL F Plus CONNECTION MANUAL (FUNCTION) 10.1 AUXILIARY FUNCTION/2ND AUXILIARY FUNCTION".

Auxiliary function during axis movement output signal MVAF1<Fn316.4>, MVAF2<Fn316.5>

[Classification] Output signal

[Function] Indicates that auxiliary functions specified at G50.9 blocks are outputting.
1st output point : MVAF1
2nd output point : MVAF2

[Output cond.] These signals turn to "1" when

- The absolute coordinate reach the 1st output point and auxiliary function code signals M00 to M31<Fn010 to Fn013> or 2nd auxiliary function code signals B00 to B31<Fn030 to Fn033> are output.
- The absolute coordinate reach the 2nd output point and auxiliary function code signals M00 to M31<Fn010 to Fn013> or 2nd auxiliary function code signals B00 to B31<Fn030 to Fn033> are output.

These signals turn to "0" when

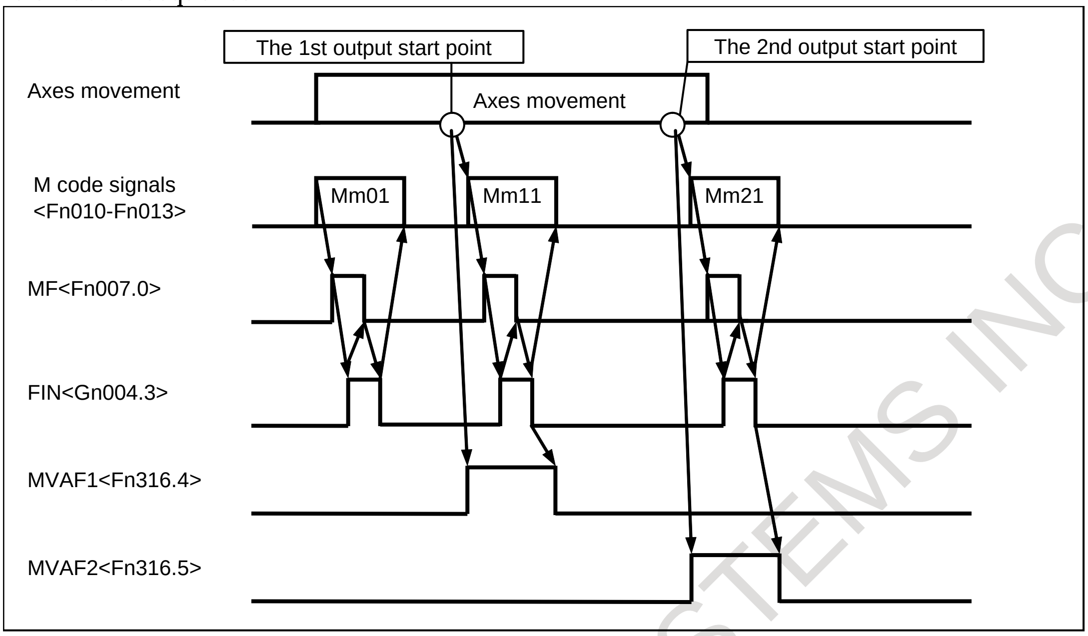
- Each FIN processing has been completed.
- Reset occurs

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn004				MFIN3	MFIN2	FIN		
Gn005	BFIN	AFL			TFIN	SFIN		MFIN

	#7	#6	#5	#4	#3	#2	#1	#0
Fn007	BF				TF	SF		MF
Fn008			MF3	MF2				
Fn010	M07	M06	M05	M04	M03	M02	M01	M00
Fn011	M15	M14	M13	M12	M11	M10	M09	M08
Fn012	M23	M22	M21	M20	M19	M18	M17	M16
Fn013	M31	M30	M29	M28	M27	M26	M25	M24
Fn014	M207	M206	M205	M204	M203	M202	M201	M200
Fn015	M215	M214	M213	M212	M211	M210	M209	M208
Fn016	M307	M306	M305	M304	M303	M302	M301	M300
Fn017	M315	M314	M313	M312	M311	M310	M309	M308
Fn030	B07	B06	B05	B04	B03	B02	B01	B00
Fn031	B15	B14	B13	B12	B11	B10	B09	B08
Fn032	B23	B22	B21	B20	B19	B18	B17	B16
Fn033	B31	B30	B29	B28	B27	B26	B25	B24
Fn316			MVAF2	MVAF1				

Example of ladder sequence



Alarm and message

Number	Message	Description
PS5330	G50.9 FORMAT ERROR	- There is not coordinates value specification in G50.9 block. - There is not M code, B code command in the G50.9 block. - G50.9 is commanded in canned cycle mode. - 3 blocks of G50.9 block are commanded consecutively.
PS5331	ILLEGAL COMMANDED POSITION	It didn't reach a commanded absolute coordinate value. The movement command or absolute coordinates at G50.9 block are wrong.

Note

Tool nose radius compensation/Cutter compensation C

Because G50.9 block is treated as a block without movement, “Tool nose radius compensation” and “Cutter compensation C” are temporarily canceled when two G50.9 blocks are commanded consecutively.

Tool compensation

When tool compensation amount is changed by T code after the G50.9 block, absolute coordinate value changes according to tool compensation amount. Therefore, absolute coordinate value sometimes enters the area specified by G50.9 block, at the head of the movement block.

Program restart

The histories of M code displayed in the screen of program restart are not in actual output order but in commanded block order.

Multiple M commands in a single block

The parameter M3B (No.3404#7) must be set to 1 in order to command multiple M codes in one block.

M30, M02, M00, M01

Decode signals DM30, DM02, DM01, DM00 are not output while axes are moving if M30, M02, M01, M00 are specified at G50.9 block. These decode signals are output at movement completion, but not output if FIN processing for M30, M02, M01, M00 have already completed.

M98, M198

Subprogram call M code (M98, M198) cannot be commanded at G50.9 block.
The alarm PS0076, "PROGRAM NOT FOUND" occurs.

Macro call M code

Custom macro call M code cannot be commanded at G50.9 block.

Tool life management, Tool management

Even if the M code (M06, etc.) which counts tool life is commanded at G50.9 block, tool life count isn't done.

Waiting M code

Even if a waiting M code is commanded at the G50.9 block, waiting function isn't executed. It is recognized as a usual auxiliary function.

Rigid tapping beginning M code

Even if M code (M29) for beginning rigid tap is commanded at G50.9 block, a mode doesn't switch over to rigid tapping.

M code group check

If plural M codes of one group are commanded at G50.9 block, the alarm PS5016, "ILLEGAL COMBINATION OF MCODES" occurs.

Manual handle retrace

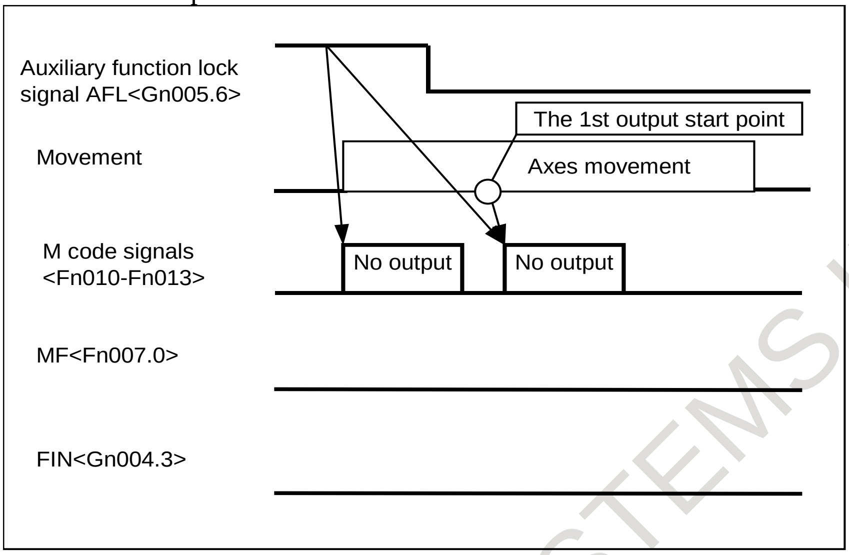
G50.9 block and movement block cannot move backward in manual handle retrace mode.
Only forward movement is possible.

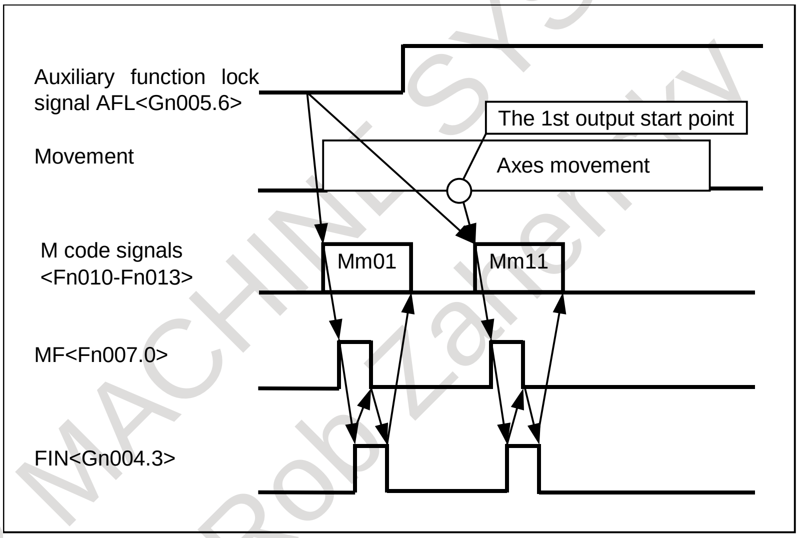
Canned cycle

Canned cycle cannot be specified as movement command block.

Auxiliary function lock

According to the condition of Auxiliary function lock signal AFL <Gn005.6> at beginning of movement block, auxiliary functions are output to PMC.





M code for spindle command (M03/M04/M05)

M code for spindle command (M03/M04/M05) is cannot be specified in G50.9 block.
These M codes are recognized as usual auxiliary functions.

M code that is set by parameter

M code in which specific operation is enabled by parameter setting is cannot be specified in G50.9 block.
This M code is recognized as a usual auxiliary function.

Retrace (M system)

G50.9 block and movement block cannot move backward. Only forward movement is possible.

Dynamic graphic display

Instruction information of M code that is specified in G50.9 block is not displayed in Path Drawing screen/Animation screen

Manual intervention

Do not operate manual intervention in G50.9 block.

Single block

During single block operation, G50.9 block stops.

Incremental programming

Incremental programming cannot be specified in G50.9 block. The alarm PS0009, ”IMPROPER NC ADDRESS” occurs.
Also, in case of incremental mode (G91) in G code system B/C, incremental programming in G50.9 block is treated as absolute programming.

10 SPINDLE SPEED FUNCTION

10.1 SPINDLE SPEED FUNCTION (S CODE OUTPUT)

Overview

When up to five digits are specified after address S, code and strobe signals are sent out and used to control the spindle speed. The code signals are retained until another S code is issued.
One S code is used for each block. Parameter No. 3031 can be used to specify the maximum number of digits. If a number greater than the maximum number is specified, an alarm can be raised.

Signal

Refer to section "Auxiliary Function/Second Auxiliary Function."

Parameter

3031	Allowable number of digits for the S code

[Data type] Byte
[Valid data range] 1 to 5
Set the allowable numbers of digits for the S code.

Note

NOTE
1 When a move command and a spindle-speed function command are specified within the same block, the commands are executed in either of the following two ways:
(1) The move command and spindle-speed function are started at the same time.
(2) After the move command is completed, the spindle-speed function command is started.
Whether (1) or (2) takes place depends on the processing on the PMC side.
2 For S code output when spindle serial output or spindle analog output is used, see Section "SPINDLE SPEED CONTROL".

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (this manual)	Auxiliary function/2nd auxiliary function
	Spindle speed control

10.2 SPINDLE SERIAL OUTPUT

Overview

Spindle serial output is a spindle motor control interface.
The spindle serial output can control up to eight serial spindles (up to four serial spindles per path).

Explanation

The terms "logical n-th spindle", "n-th spindle in a path", and "n-th spindle amplifier" in the subsequent descriptions refer to the following:

- Logical n-th spindle:
  The n-th spindle to be controlled in the overall system
- N-th spindle (in a path):
  The n-th spindle to be controlled in a path.
  The first, second, third, and fourth spindles in the descriptions refer to relative spindle numbers within a path. (For a single-path system, the logical n-th spindle equals the n-th spindle.)
- N-th spindle amplifier:
  The n-th spindle amplifier (amplifier No. n) actually connected to the CNC.

For the relationships between them, see "Spindle numbers vs. spindle motors" described later.

For how to set a spindle configuration for each path in multipath control, see the description of spindles in "Multipath Control".
The relationship between the spindles and functions is as shown Table 10.2 (a).

Table 10.2 (a)

Spindle / Function	Serial spindle (*1)			
	First spindle	**Second spindle**	**Third spindle**	**Fourth spindle**
Threading/feed per revolution (synchronous feed)	Available	Available(*3)	Available(*3)	Available(*3)
Constant surface spindle control	Available	Available(*3)	Available(*3)	Available(*3)
Spindle speed fluctuation detection (T series)	Available	Available(*3)	Available(*3)	Available(*3)
Actual spindle speed output (T series)	Available	Available(*3)	Available(*3)	Available(*3)
Spindle positioning (T series)	Available	Available(*3)	Available(*3)	Available(*3)
Cs contour control	Available	Available	Available	Available
Multi-spindle control	Available	Available	Available	Available
Rigid tapping	Available	Available(*3)	Available(*3)	Available(*3)
Spindle synchronous control	Available	Available	Available	Available
Spindle command synchronous control	Available	Available	Available	Available
Spindle control unit functions, such as spindle orientation, spindle output switching, spindle switching, and etc. (*4)	Available	Available	Available	Available
Polygon turning (T series) (using the servo motor axis and spindle)	Available	Available(*3)	Available(*3)	Available(*3)
Polygon machining with two spindles (T series)	Available	Available	Available	Available
Spindle output control by the PMC	Available	Available	Available	Available

NOTE
1 Enable spindle serial output (bit 5 (SSN) of parameter No.8133 is 0).
2 The number of controlled spindles must be specified.
3 Enable multi-spindle control (bit 3 (MSP) of parameter No.8133 is 1).
4 These functions belong to the spindle control unit. They cannot be used unless the spindle control unit supports those functions.

- Spindle numbers vs. spindle motors

Logical spindle numbers are associated with spindle motors as follows:

1. Logical spindle numbers and amplifier numbers
   Logical spindle numbers indicate the order of spindles to be controlled (the logical arrangement order). Parameters for the individual spindles are arranged in this order. Logical spindle numbers are assigned sequentially from 1 to the maximum number of spindles. DI/DO signals are allocated for each spindle number within a path, and their signal locations are incremented by 1000 for each path.
   Amplifier numbers indicate the order of spindle amplifiers actually connected to the CNC (the physical arrangement order). They are assigned as shown below (Fig. 10.2 (a)). When a spindle motor is connected to each spindle amplifier, the spindle amplifier numbers and spindle motor numbers match.

2. Setting method

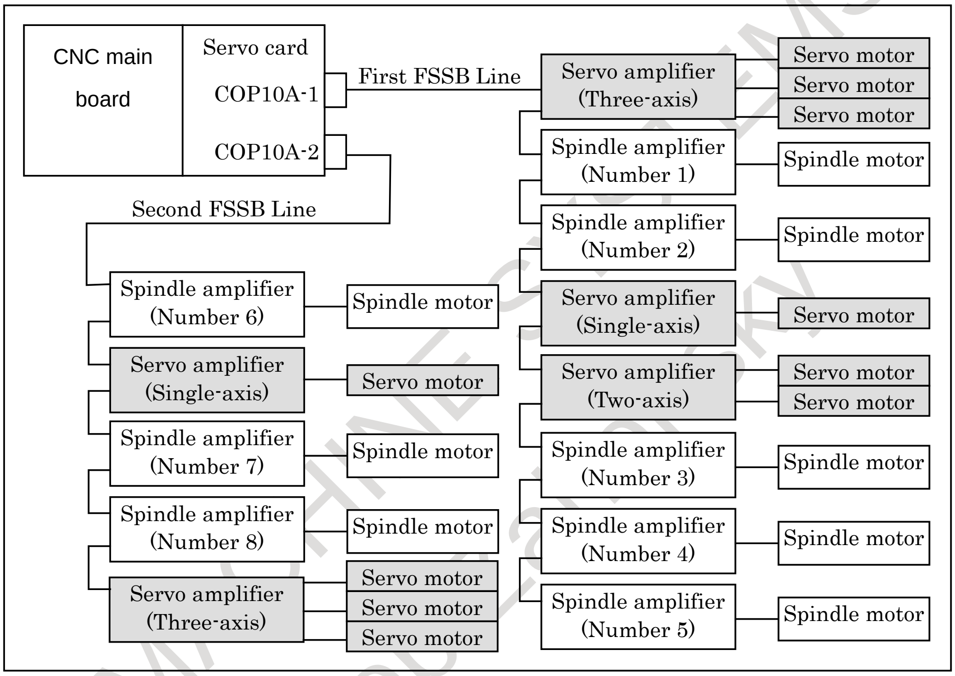


Fig. 10.2 (a) Connection of serial spindles

For each spindle, set the type of the motor in bit 0 (A/S) of parameter No. 3716, and set the amplifier number in parameter No. 3717. For spindles not to be used, set 0 as the amplifier numbers.

Example 1) When using one serial spindle in a single-path system

Parameter	First spindle	Second spindle	Third spindle	Fourth spindle
Bit 0 (A/S) of parameter No. 3716	1	0	0	0
No.3717	1	0	0	0

Example 2) When using four serial spindles in a single-path system

Parameter	First spindle	Second spindle	Third spindle	Fourth spindle
Bit 0 (A/S) of parameter No. 3716	1	1	1	1
No.3717	1	2	3	4

NOTE
In addition, it is necessary to set FSSB. Refer to "1.4.4 FSSB Setting" for details.

Signal

Spindle control unit signals for the serial spindle

<Gn070 - Gn073><Gn304 - Gn307><Fn045 - Fn048><Fn306 - Fn307> : 1st spindle
<Gn074 - Gn077><Gn308 - Gn311><Fn049 - Fn052><Fn308 - Fn309> : 2nd spindle
<Gn204 - Gn207><Gn312 - Gn315><Fn168 - Fn171><Fn310 - Fn311> : 3rd spindle
<Gn266 - Gn269><Gn316 - Gn319><Fn266 - Fn269><Fn312 - Fn313> : 4th spindle

These addresses are assigned to locations on the CNC, but the signals at these addresses are input/output signals of the serial spindle control unit. For details of the signals at these addresses, refer to the following manuals of serial spindles:

FANUC SERVO AMPLIFIER α*i*-B series DESCRIPTIONS (B-65412EN)

FANUC AC SPINDLE MOTOR α*i*-B / β*i*-B series DESCRIPTIONS (B-65452EN)

Serial spindle control unit signals (such as the alarm signals ALMA, ALMB <F045.0, F049.0> etc) are disabled and its operation is not guaranteed after power-on until the parameters have been transferred from the CNC to the spindle amplifier.

After checking the following spindle operation ready signals to see the ready status, use serial spindle control unit signals.

Spindle ready signals

SRSP1R<Fn034.6> : 1st spindle
SRSP2R<Fn034.5> : 2nd spindle
SRSP3R<Fn034.4> : 3rd spindle
SRSP4R<Fn034.3> : 4th spindle

[Classification] Output signal

[Function] These signals post that a corresponding spindle is ready for operation.

[Operation] These signals are set to “1” when:

The corresponding spindle becomes ready for operation (when the following conditions are met).

- The option for the corresponding spindle is provided and is set for use.
- There is no serial communication alarm. (Serial spindle)
- Serial spindle parameters have been transferred. (Serial spindle)
- The spindle control software is ready for operation. (Serial spindle)

These signals are set to “0” when:

The corresponding spindle is not ready for operation.

All-spindle operation ready signal SRSRDY <F0034.7>

[Classification] Output signal

[Function] This signal posts that all spindles used are ready for operation.

[Operation] The signal is set to “1” when:

All spindles used become ready for operation.

The signal is set to “0” when:

Some of the spindles used are not ready for operation.

Spindle warning detail signals SPWRN1 to SPWRN9 <Fn264.0 to Fn265.0>

[Classification] Output signal

[Function] Before an alarm is issued, the occurrence of a warning condition is posted via the warning interface, which is independent of the conventional alarm interface.

Nine signals are output to indicate a decimal code. The relationship between the spindle warning number and signals is expressed as follows:

[Operation] A warning number is output according to the spindle amplifier status as follows:

$$\text{Spindle warning No.} = \sum_{i=1}^{8}\left\{2^{i-1}\times SPWRN_i\right\} + 2^8\times SPWRN_9$$

Warning No.	Description of warning	Spindle operation
56	SPM internal cooling fan stopped	When the stopped state of the internal cooling fan is detected, the warning number is posted from the spindle amplifier to the CNC. The spindle continues operating. About one minute later, an alarm number is posted, and the spindle is placed in the free running state.
88	SPM radiator cooling fan stopped	When the stopped state of the external cooling fan is detected, the warning number is posted from the spindle amplifier to the CNC. The spindle continues operating. If the main circuit overheats because the external cooling fan is stopped, an alarm is posted, and the spindle is placed in the free running state.
58	Overload on converter main circuit	A warning number is output according to the signal received from the PSM In the warning state, the spindle continues operating.
59	Converter internal cooling fan stopped	
113	Converter radiator cooling fan stopped	

The spindle warning detail signals are set to “0” when all alarm factors are eliminated.
When a warning is issued in more than one spindle within the same path, the spindle warning detail signals indicate the warning No. of the spindle having the smallest spindle number.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F0034	SRSRDY							

	#7	#6	#5	#4	#3	#2	#1	#0
Fn034		SRSP1R	SRSP2R	SRSP3R	SRSP4R			

	#7	#6	#5	#4	#3	#2	#1	#0
Fn264	SPWRN8	SPWRN7	SPWRN6	SPWRN5	SPWRN4	SPWRN3	SPWRN2	SPWRN1
Fn265								SPWRN9

- Serial spindle control unit signals

For first spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn070	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
Gn071	RCHA	RSLA	INTGA	SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
Gn072	RCHHGA	MFNHGA	INCMDA	OVRA	DEFMDA	NRROA	ROTAA	INDXA
Gn073						MPOFA	SLVA	MORCMA
Gn304								
Gn305								
Gn306								
Gn307								
Fn045	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
Fn046				SLVSA	RCFNA	RCHPA	CFINA	CHPA
Fn047							INCSTA	PC1DTA
Fn048								
Fn306								
Fn307								

For second spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn074	MRDYB	ORCMB	SFRB	SRVB	CTH1B	CTH2B	TLMHB	TLMLB
Gn075	RCHB	RSLB	INTGB	SOCNB	MCFNB	SPSLB	*ESPB	ARSTB
Gn076	RCHHGB	MFNHGB	INCMDB	OVRB	DEFMDB	NRROB	ROTAB	INDXB
Gn077						MPOFB	SLVB	MORCMB
Gn308								
Gn309								
Gn310								
Gn311								
Fn049	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB
Fn050				SLVSB	RCFNB	RCHPB	CFINB	CHPB
Fn051							INCSTB	PC1DTB
Fn052								
Fn308								
Fn309								

For third spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn204	MRDYC	ORCMC	SFRC	SRVC	CTH1C	CTH2C	TLMHC	TLMLC
Gn205	RCHC	RSLC	INTGC	SOCNC	MCFNC	SPSLC	*ESPC	ARSTC
Gn206	RCHHGC	MFNHGC	INCMDC	OVRC	DEFMDC	NRROC	ROTAC	INDXC
Gn207						MPOFC	SLVC	MORCMC
Gn312								
Gn313								
Gn314								
Gn315								
Fn168	ORARC	TLMC	LDT2C	LDT1C	SARC	SDTC	SSTC	ALMC
Fn169				SLVSC	RCFNC	RCHPC	CFINC	CHPC
Fn170							INCSTC	PC1DTC
Fn171								
Fn310								
Fn311								

For fourth spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn266	MRDYD	ORCMD	SFRD	SRVD	CTH1D	CTH2D	TLMHD	TLMLD
Gn267	RCHD	RSLD	INTGD	SOCND	MCFND	SPSLD	*ESPD	ARSTD
Gn268	RCHHGD	MFNHGD	INCMDD	OVRD	DEFMDD	NRROD	ROTAD	INDXD
Gn269						MPOFD	SLVD	MORCMD
Gn316								
Gn317								
Gn318								
Gn319								
Fn266	ORARD	TLMD	LDT2D	LDT1D	SARD	SDTD	SSTD	ALMD
Fn267				SLVSD	RCFND	RCHPD	CFIND	CHPD
Fn268							INCSTD	PC1DTD
Fn269								
Fn312								
Fn313								

Parameter

- Settings for spindle motors and spindle numbers

	#7	#6	#5	#4	#3	#2	#1	#0
3716								A/Ss

[Input type] Parameter input
[Data type] Bit spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 A/Ss Spindle motor type is :
0: Analog spindle.
1: Serial spindle.

> **NOTE**
> 1 To use a serial spindle, set 0 to bit 5 (SSN) of parameter No.8133.
> 2 The 1 analog spindle at the maximum can be controlled in a system.
> 3 When using an analog spindle, assign it at the end of the spindle configuration.

3717	Motor number to each spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled axes

Set a spindle amplifier number to be assigned to each spindle.
0: No spindle amplifier is connected.
1: Spindle motor connected to amplifier number 1 is used.
2: Spindle motor connected to amplifier number 2 is used.
to
n: Spindle motor connected to amplifier number n is used.

NOTE
1 In addition, it is necessary to set FSSB. Refer to "1.4.4 FSSB Setting" for details.
2 If value of this parameter is larger than the maximum value, the alarm SP1996, "ILLEGAL SPINDLE PARAMETER SETTING" occurs.
3 If the spindle motor is treated as hypothetical Cs axis control or spindle control with servo motor, set this parameter to “0”.
4 In the display order of the parameter No.982, the spindle axes since the spindle axis that 0 is set to this parameter become invalid, too. However, the case that the spindle motor is treated as hypothetical Cs axis control or spindle control with servo motor is excepted.
Example) On the following setting, S3 and S4 spindle axes are invalid when the spindle motor of S3 is not treated as hypothetical Cs axis control or spindle control with servo motor.

Display order of No.982	Setting value of No.982	Setting value of No.3717
S1	1	1
S2	2	2
S3	1	0
S4	2	3

3718	Subscript for display of a serial spindle (main spindle) or analog spindle

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 122

Set a subscript to be added to spindle speed display on a screen such as the position display screen.
Used when the spindle switching function is not used, or used for the main spindle when the spindle switching function is used.

NOTE
This parameter is invalid when an extended spindle name is used.

3719	Subscript for display of a serial spindle (sub-spindle)

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 122

Set a subscript to be added to spindle speed display on a screen such as the position display screen.
Used for the sub-spindle when the spindle switching function is used.

NOTE
This parameter is invalid when an extended spindle name is used.

- Connection of serial spindle control unit

	#7	#6	#5	#4	#3	#2	#1	#0
4019	PLD							
4195	PLD							

[Input type] Parameter input
[Data type] Bit spindle

#7 PLD When power is turned on, spindle amplifier parameters for the serial spindle are:
0: Not set automatically.
1: Set automatically.
No. 4019 : (Used when the spindle switching function is not used, or used for the main spindle when the spindle switching function is used)
No. 4195 : (Used for the sub-spindle when the spindle switching function is used)
When this parameter is set to 1 after the spindle motor model code parameter is set, the next power-on operation sets the standard values for the motor model in parameters and sets this parameter to 0.

4133	Spindle motor model code (Used when spindle switching function is not used, or used for main spindle in spindle switching)
4309	Spindle motor model code (For sub-spindle when spindle switching function is provided)

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767
When setting initial parameters for the serial spindle, specify the spindle motor model code.

- Parameters for the serial spindle control unit

Parameters Nos. 4000 to 4799: For serial spindles
Although the above parameters are stored as CNC parameters, they are actually used by the spindle control unit of serial spindles.
For details of these parameters, refer to the following manuals of serial spindles:
FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)

Alarm and message

Number	Message	Description
SP1220	NO SPINDLE AMP.	Either the cable connected to a serial spindle amplifier is broken, or the serial spindle amplifier is not connected.
SP1227	RECEIVING ERROR (SERIAL SPINDLE)	A receive error occurred in communications between the CNC and the serial spindle amplifier.
SP1228	COMMUNICATION ERROR (SERIAL SPINDLE)	A communications error occurred between the CNC and the serial spindle amplifier.
SP1229	COMMUNICATION ERROR SERIAL SPINDLE AMP.	A communications error occurred between serial spindle amplifiers (amplifiers of even number and odd number).
SP1245 to SP1247	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC. Inquire of FANUC, please.
SP1969 to SP1972	SPINDLE CONTROL ERROR	An error occurred in the spindle control software. Inquire of FANUC, please.

Number	Message	Description
SP1976 to SP1979	SERIAL SPINDLE COMMUNICATION ERROR	An error occurred in the spindle control software. Inquire of FANUC, please.
SP1980 to SP1984	SERIAL SPINDLE AMP. ERROR	Defect on serial spindle amplifier. Inquire of FANUC, please.
SP1985 to SP1987	SERIAL SPINDLE CONTROL ERROR	Defective SIC–LSI on serial spindle amplifier. Inquire of FANUC, please.
SP1988, SP1989	SPINDLE CONTROL ERROR	An error occurred in the spindle control software. Inquire of FANUC, please.
SP1996	ILLEGAL SPINDLE PARAMETER SETTING	The spindle was assigned incorrectly. Check to see the following parameter. (No.3716 or 3717)
SP9000 or later		An alarm is issued from the spindle amplifier unit of the serial spindle. For details, refer to the manual of the serial spindle.

Diagnosis data

- Information about spindle control

	#7	#6	#5	#4	#3	#2	#1	#0
1570	LNK							

[Data type] Bit spindle

LNK 0: The spindle interface does not operate normally.
1: The spindle interface operates normally.

403	Spindle motor temperature information

[Data type] Byte spindle
[Unit of data] °C
[Valid data range] 0 to 255

When the α*i* spindle interface is used, the temperature of the αi spindle motor winding is indicated. The overheat temperature varies depending on the motor.

> **NOTE**
> The overheat temperature has the following error:
> Lower than 160°C: Up to 5°C
> 160 to 180°C: Up to 10°C

- Indication of serial spindle load meter and speed meter

410	Spindle load meter indication (%)

411	Spindle motor speed indication (min^{-1})

[Data type] Word spindle

For correct indication of the load meter and motor speed, the following parameters must be set properly:
Maximum motor speed: Set for each axis in parameters Nos. 4020 (MAIN) and 4196 (SUB).
Load meter indication at maximum output: Set for each axis in parameters Nos. 4127 (MAIN) and 4276 (SUB).

> **NOTE**
> Parameters for (MAIN) and (SUB) must be set when the spindle switching function is used.
> When the spindle switching function is not used, set the parameters for (MAIN).

445	Position data of position coder (pulse)

[Data type] 2-word spindle
[Unit of data] Pulse
[Valid data range] Depend on the bit 0 (ORPUNT) of No.4542 setting (Refer to the table below)

Bit 0 (ORPUNT) of No.4542	Valid data range
0	0 to 4095
1	0 to 32767

> **NOTE**
> Once spindle orientation is performed after power-on, position data can be read. To read position data, set bit 1(SPP) of parameter No. 3117 to 1.
> (Except when maintenance is performed, bit 1(SPP) of parameter No. 3117 must be set to 0.)

449	Position data of position coder (angle)

[Data type] Real spindle
[Unit of data] Refer to the table below
[Valid data range] 0 to 359.999[deg]

Diagnosis data No.449 indicates spindle angle from the one-rotation signal when bit 1 (SPP) of parameter No.3117 is set to 1. Spindle angle displays to three decimal places. It is available for serial spindle. To display spindle angle, spindle orientation must be performed once. Unit of spindle angle data is as follows.

Bit 0 (ORPUNT) of No.4542	Unit of data [deg]
0	0.088
1	0.011

> **NOTE**
> Once spindle orientation is performed after power-on, position data can be read. To read position data, set bit 1(SPP) of parameter No. 3117 to 1.
> (Except when maintenance is performed, bit 1(SPP) of parameter No. 3117 must be set to 0.)

710	Spindle status error No.

[Data type] Byte spindle
The status error number transferred from the spindle amplifier is indicated.

When the spindle is disabled by an error such as a PMC signal input error (for example, when two operation modes are specified at the same time), a status error is recognized while the spindle is disabled. When the signal is input normally, the normal operation is resumed.

For details of spindle error numbers, see the description of alarms (SP alarms) related to serial spindles in the alarm list.

712	Spindle warning No.

[Data type] Byte spindle
The warning number transferred from the spindle amplifier is indicated.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	FSSB Setting

10.3 SPINDLE ANALOG OUTPUT

Overview

Spindle analog output is a spindle motor control interface.
The spindle analog output can control up to 1 analog spindle.

This section assumes that an analog spindle is set for the first spindle (serial spindles are not used with an analog spindle, but only an analog spindle is used). When using both serial and analog spindles, see the next section, "SERIAL/ANALOG SPINDLE CONTROL".

Explanation

The terms "logical n-th spindle", "n-th spindle in a path", and "n-th spindle amplifier" used herein are defined as follows:

- Logical n-th spindle:
  The n-th spindle to be controlled as counted throughout the system
- N-th spindle (in a path):
  The n-th spindle to be controlled as counted within a path
  The terms "first spindle", "second spindle", "third spindle", and "fourth spindle" used herein refer to relative spindle numbers within a path. (In a single-path system, the logical n-th spindle is the n-th spindle.)

For the relationships between them, see "Spindle numbers vs. spindle motors" described later.

For how to set a spindle configuration for each path to be subjected to multi-path control, see the description of spindles in "Multipath Control".

The Table 10.3 (a) lists the relationships between the spindles and functions.

Table 10.3 (a)

Spindle / Function	Analog spindle (*1)
	First spindle
Threading/feed per revolution (synchronous feed)	Available
Constant surface speed control	Available
Spindle speed fluctuation detection (T series)	Available
Actual spindle speed output (T series)	Available
Spindle positioning (T series)	Available

Function \ Spindle	Analog spindle (*1)
	First spindle
Cs contour control	Unavailable
Multi-spindle control	Unavailable
Rigid tapping	Available
Spindle synchronous control	Unavailable
Spindle simple synchronous control	Unavailable
Spindle control unit functions, such as spindle orientation, spindle output switching, and spindle switching (*2)	Available
Polygon machining (T series) (using the servo motor axis and spindle)	Available
Polygon machining with two spindles (T series)	Unavailable
Spindle output control by the PMC	Available

NOTE
1 These functions belong to the spindle control unit. They cannot be used unless the spindle control unit supports them.
2 The number of controlled spindles must be specified.

- Spindle numbers vs. spindle motors

Logical spindle numbers are associated with spindle motors as follows:

1. Logical spindle numbers and amplifier numbers
   Logical spindle numbers indicate the order of spindles to be controlled (the logical arrangement order). Parameters for the individual spindles are arranged in this order. Logical spindle numbers are assigned sequentially from 1 to the maximum number of spindles.
   Amplifier numbers indicate the order of spindle amplifiers actually connected to the CNC (the physical arrangement order). They are assigned as shown below (Fig. 10.3 (a)).
   (Because only one analog spindle can be connected, the spindle amplifier and spindle motor are each assigned with number 1.)

2. Setting method

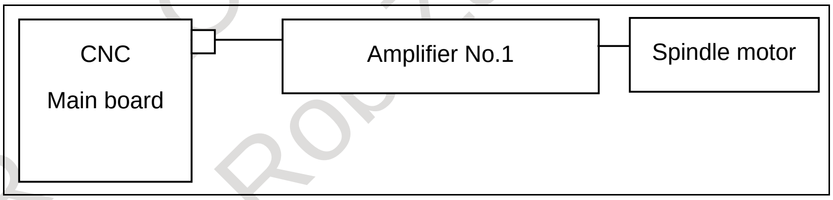


Fig. 10.3 (a) Connection of analog spindle

3. Setting
   For each spindle, the type of the motor used is set in bit 0 (A/S) of parameter No. 3716, and the amplifier number, in parameter No. 3717. For spindles not to be used, the amplifier numbers are 0.

Parameter	First spindle	Second spindle	Third spindle	Fourth spindle
Bit 0 (A/S) of parameter No. 3716	0	0	0	0
No.3717	1	0	0	0

Note

NOTE
1 If a failure occurs in the D/A converter, the following conditions occur.
- Alarm SP1241, “D/A CONVERTER ERROR” is issued.
- The spindle command voltage is set to 0.
- The spindle enable signal is set to “0”.

2 The load meter value (motor load) always becomes 0 no matter what condition occurs.

Parameter

- Settings for spindle motors and spindle numbers

	#7	#6	#5	#4	#3	#2	#1	#0
3716								A/Ss

[Input type] Parameter input
[Data type] Bit spindle

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 A/Ss Spindle motor type is :
0: Analog spindle.
1: Serial spindle.

NOTE
When a serial spindle is used, enable spindle serial output (bit 5 (SSN) of parameter No.8133 is 0).

3717	Motor number to each spindle

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled axes
Set a spindle amplifier number to be assigned to each spindle.
0: No spindle amplifier is connected.
1: Spindle motor connected to amplifier number 1 is used.
2: Spindle motor connected to amplifier number 2 is used.
to
n: Spindle motor connected to amplifier number n is used.

3718	Subscript for display of a serial spindle (main spindle) or analog spindle

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 122
Set a subscript to be added to spindle speed display on a screen such as the position display screen.

Used when the spindle switching function is not used, or used for the main spindle when the spindle switching function is used.

> **NOTE**
> This parameter is invalid when an extended spindle name is used.

	#7	#6	#5	#4	#3	#2	#1	#0
3799							NDPs	

[Input type] Parameter input
[Data type] Bit spindle

#1 NDPs When an analog spindle is used, a position coder disconnection check is:
0: Made.
1: Not made.
(This parameter is valid when bit 0 (NAL) of parameter No. 3799 is set to 0.)
When no position coder is used with an analog spindle, set this parameter to 1.

Alarm and message

Number	Message	Description
SP1241	D/A CONVERTER ERROR	The D/A converter for controlling analog spindles is erroneous.
SP1996	ILLEGAL SPINDLE PARAMETER SETTING	The spindle was assigned incorrectly. Check to see the following parameter. (No.3716 or 3717)

10.4 SERIAL/ANALOG SPINDLE CONTROL

Overview

Serial/analog spindle control allows an analog spindle to be used as one of the controlled spindles in the entire system. Only spindle speed command control and spindle speed command control by the PMC can be used.

> **NOTE**
> 1 When using a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0.
> 2 The analog spindle must be assigned to the last of the spindle configuration.

Explanation

Serial/analog spindle control allows only one analog spindle to be used as one of the controlled spindles in the entire system. The analog spindle used must be assigned to the last spindle of the spindle configuration.

Spindle functions available in serial/analog spindle control

The table below lists the relationship between the spindles and functions:

Function \ Spindle	Serial spindle (NOTE 1)	Analog spindle
Threading/feed per revolution (synchronous feed)	Available	Unavailable
Constant surface speed control	Available	Unavailable
Spindle speed fluctuation detection	Available	Unavailable
Actual spindle speed output (T series)	Available	Unavailable
Spindle positioning (T series)	Available	Unavailable
Cs contour control	Available	Unavailable

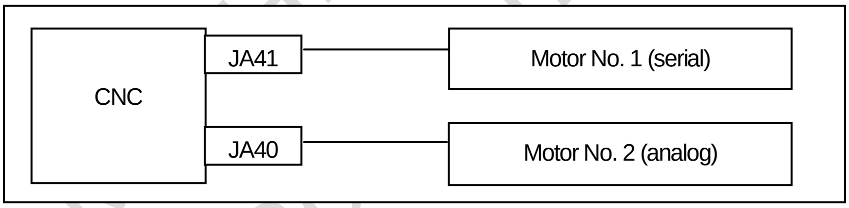
Spindle / Function	Serial spindle (NOTE 1)	Analog spindle
Rigid tapping	Available	Unavailable
Spindle synchronous control	Available	Unavailable
Simple spindle synchronous control	Available	Unavailable
Spindle control unit functions, such as spindle orientation, spindle output switching, and spindle switching (NOTE 2)	Available	Available
Polygon machining (T series) (using the servo motor axis and spindle)	Available	Unavailable
Polygon machining with two spindles (T series) (Spindle and spindle polygon)	Available	Unavailable
Spindle output control by the PMC	Available	Available

NOTE
1 When using a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0.
2 These functions belong to the spindle control unit. They cannot be used unless the spindle control unit supports these functions.

Connection example

Connect the analog spindle to JA40.
When an analog spindle is used under serial/analog spindle control, a position coder cannot be attached to the analog spindle.
For connection of serial spindles, refer to CONNECTION MANUAL (Hardware) (B-64603EN).

- When connecting one serial spindle and one analog spindle (when the maximum number of controllable spindles is two)



Restrictions

When an analog spindle is used under serial/analog spindle control, there are restrictions as follows:

1 Be sure to assign the analog spindle as the last spindle of the controlled spindles.
Example:
When there are two controlled spindles, and one of these spindles is an analog spindle
The controlled spindle number of the analog spindle is 2.
Set the spindle amplifier number of the analog spindle (parameter No. 3717) to 2.
2 A position coder cannot be installed.
3 Only spindle speed command control and spindle speed command control by the PMC can be used. Other control and spindle functions cannot be used.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3702							EMS	

[Input type] Parameter input
[Data type] Bit path

#1 EMS The multi-spindle control is:
0: Used.
1: Not used.

> **NOTE**
> Make the setting on the side of the path in which multi-spindle control is unnecessary in 2-path control.

	#7	#6	#5	#4	#3	#2	#1	#0
3716								A/Ss

[Input type] Parameter input
[Data type] Bit spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 A/Ss Spindle motor type is :
0: Analog spindle.
1: Serial spindle.

> **NOTE**
> 1 A maximum of one analog spindle can be controlled.
> 2 To use a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0.
> 3 When using an analog spindle, assign it at the end of the spindle configuration.

3717	Motor number to each spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled axes

Set a spindle amplifier number to be assigned to each spindle.
0: No spindle amplifier is connected.
1: Spindle motor connected to amplifier number 1 is used.
2: Spindle motor connected to amplifier number 2 is used.
3: Spindle motor connected to amplifier number 3 is used.

> **NOTE**
> When using an analog spindle, assign it at the end of the spindle configuration.
> (Example)
> When there are three spindles in an entire system (two serial spindles and one analog spindle), set the spindle amplifier number (this parameter) of the analog spindle to 3.

3718	Subscript for display of a serial spindle or analog spindle

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 122
Set a subscript to be added to spindle speed display on a screen such as the position display screen.

	#7	#6	#5	#4	#3	#2	#1	#0
3798								ALM

[Input type] Parameter input
[Data type] Bit

#0 ALM The spindle alarm (SP****) for all spindles is:
0: Enabled.
1: Ignored.
When this parameter is set to 1, the spindle-related alarms are ignored. So, be sure to set this parameter to 0 at all times except for special cases such as maintenance.

	#7	#6	#5	#4	#3	#2	#1	#0
3799							NDPs	NALs

[Input type] Parameter input
[Data type] Bit spindle

#0 NALs An alarm detected on the spindle amplifier side is:
0: Displayed.
1: Not displayed.
(This parameter is valid when bit 0 (ALM) of parameter No. 3798 is set to 0.)
When this parameter is set to 1, an alarm detected on the spindle amplifier side is ignored. So, be sure to set this parameter to 0 at all times except for special cases such as maintenance.

#1 NDPs When an analog spindle is used, a position coder disconnection check is:
0: Made.
1: Not made.
(This parameter is valid when bit 0 (NAL) of parameter No. 3799 is set to 0.)
When no position coder is used with an analog spindle, set this parameter to 1.

> **NOTE**
> When an analog spindle is used simultaneously with serial spindle by serial/analog spindle control, a position coder cannot be used as the analog spindle. Be sure to set this parameter to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
8133			SSN					

[Input type] Parameter input
[Data type] Bit

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#5 SSN Spindle serial output is:
0: Used.
1: Not Used.

Set this parameter as shown below depending on the spindle configuration.

Spindle configuration	Parameter SSN
When all spindles in the entire system are serial spindles	0
When serial spindles and analog spindles are mixed in the entire system	0
When all spindles in the entire system are analog spindles	1

10.5 SPINDLE SPEED CONTROL

Explanation

This section explains the following in relation to spindle speed control:

- Command flow in spindle speed control
- Processing for gear change (M and T type gear selection methods)
- Position coder feedback signal
- Speed arrival signal (SAR)

- Command flow of spindle speed control

The following Fig. 10.5 (a) summarizes spindle speed control.

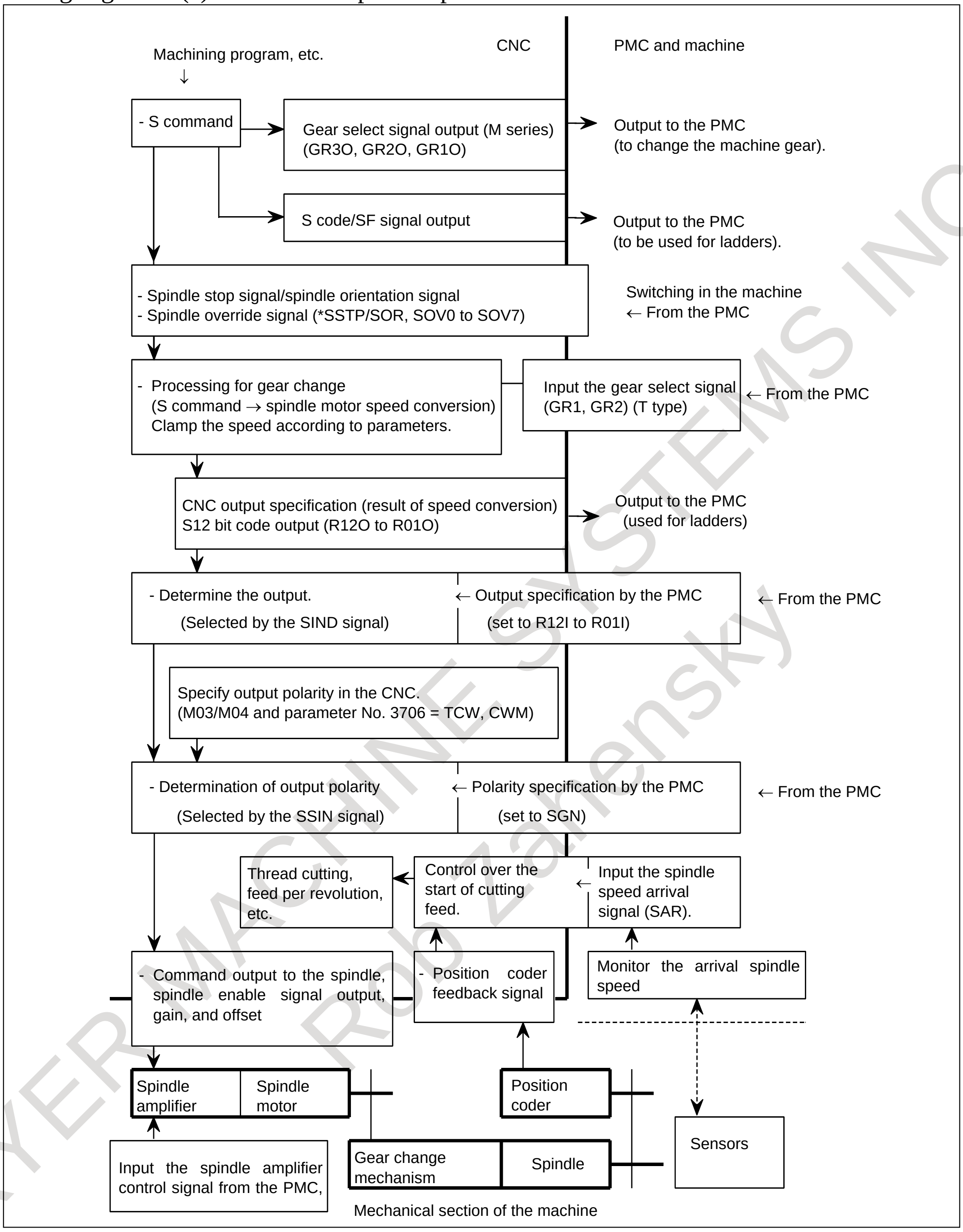


Fig. 10.5 (a)

- S command

The S command specifies the spindle speed entered from machining programs, etc. for the CNC.

For constant surface speed control (during G96 mode), the CNC converts the specified surface speed to the spindle speed.

M

When the M type gear selection method is used (a machining center is used, the constant surface speed control is not provided (bit 0 (SSC) of parameter No. 8133 is 0), and bit 4 (GTT) of parameter No. 3706 is 0), the CNC determines the necessary gear stage for obtaining a certain spindle speed according to the settings in parameters Nos. 3741, 3742, 3743, and so on and the S command value, and posts the gear stage to the PMC.
(GR3O, GR2O, GR1O<Fn034.2, Fn034.1, Fn034.0>)

- S code/SF signal output

With the spindle serial output (bit 5 (SSN) of parameter No. 8133 is 0), the spindle control function in the CNC converts the S command value to the output value for the spindle motor. For correspondence to gear change and constant surface speed control, the S code/SF signals output is different as follows in case of the spindle serial output and spindle analog output are not used.

M

M series → Outputs the S code.
The SF signal is output only when the CNC directs the PMC to change the gear.

T

T series → Outputs neither S code nor SF signal.
(This is because the S code is not always the spindle speed when the constant surface speed control is used.)

If you use the S code and SF signal for processing in the PMC ladder, you must specify parameters related to parameter No. 3705.

- Spindle stop signal (*SSTP)

When the CNC performs spindle speed control, the speed command output to the spindle is set to “0” by this signal. Setting this signal to “0” stops the spindle.
To enable speed commands to be issued to the spindle, set this signal to “1”.

- Spindle orientation signal (SOR)

If the spindle orientation signal is logical 1 and the spindle stop signal is logical 0, the spindle rotates in the direction specified by bit 5 (ORM) of parameter No. 3706 at a constant speed specified by parameter No. 3732.
Because the spindle rotates at a constant speed regardless of the gear stage, this signal can be used to rotate the spindle to drive the stopper or pin during mechanical spindle positioning.

M

For the machining center system, setting bit 1 (GST) of parameter No. 3705 enables the spindle motor to rotate at a constant speed. This function can be used for gear shifting because it maintains a constant speed of the gear change mechanism.

- Spindle speed override signal (SOV00 to SOV07)

This signal specifies an override of 0% to 254% for the specified S value for spindle control.
However, the spindle speed override function is disabled when the CNC is in the following state:
Tapping cycle (M series : G84, G74 T series : G84, G88)
Thread cutting (M series : G33 T series : G32, G92, and G76)
When the spindle speed control is performed but the spindle speed override is not used, set the override value to 100%.

- Processing for gear changing

Although the S command contains the spindle speed, the object that is actually controlled is the spindle motor. Therefore, the CNC must have some provision to detect the gear stage between the speed and spindle motor.
There are two types of gear selection methods:

M type (for the machining center system only)
The CNC selects a gear stage according to the range of speed for each gear stage previously specified in a parameter, as directed by the S command, and informs the PMC of the selected gear stage (one of the three gear stages) using the gear select signal output (GR3O, GR2O, GR1O).
Also, the CNC outputs the spindle speed based on the selected gear stage (output as the gear select signal).

T type
The gear stage to be used is determined on the machine side, and the gear stage is input using the gear selection signals (GR1 and GR2) (one of the four gear stages).
The CNC outputs the appropriate speed command for the selected gear range.

- Selection of gear change system

M

The machining center system can use either M or T type.
M type → Without constant surface speed control, and bit 4 (GTT) of parameter No. 3706 = 0
T type → With constant surface speed control, or bit 4 (GTT) of parameter No. 3706 = 1

NOTE
The M type gear selection method cannot be selected when one of the following functions is used:
- Constant surface speed control
- Multi-spindle control
- Extended spindle name
- Spindle control between paths

T

The lathe system can use only T type.

- M type gear selection method

By specifying from s0 to S99999 in memory or MDI operation, the CNC outputs a command corresponding to the spindle speed. There is a two-speed (GR1O and GR2O) or three-speed range (GR1O, GR2O, GR3O), set by parameters Nos. 3741-3743, and the gear selection signal is output simultaneously.
When the gear selection signal is changed, the SF signal is output at the same time.
By setting bit 6 (SFA) of parameter No. 3705, SF can also be output regardless of gear change.
The meaning of the gear signals is Table 10.5 (a):

Table 10.5 (a)

Gear selection signals	Gear 2-stage	Gear 3-state	Remarks
GR1O	Low	Low	Low : Low Gear Middle : Middle Gear High : High Gear
GR2O	High	Middle	
GR3O		High	

The speed commands output to the spindle motor are as follows:

- For the serial spindle, the speed commands are processed as values 0 to 16383 between the CNC and spindle control unit.
- For the analog spindle, the speed commands are output to the analog voltage signal SVC as analog voltages 0 to 10 V.

The following descriptions exemplify the analog spindle. However, they can be applied to the serial spindle on the assumption that spindle motor speed with analog voltage 10 V corresponds to the maximum spindle motor speed.

(i) M type gear change method A (See Fig. 10.5 (b).)

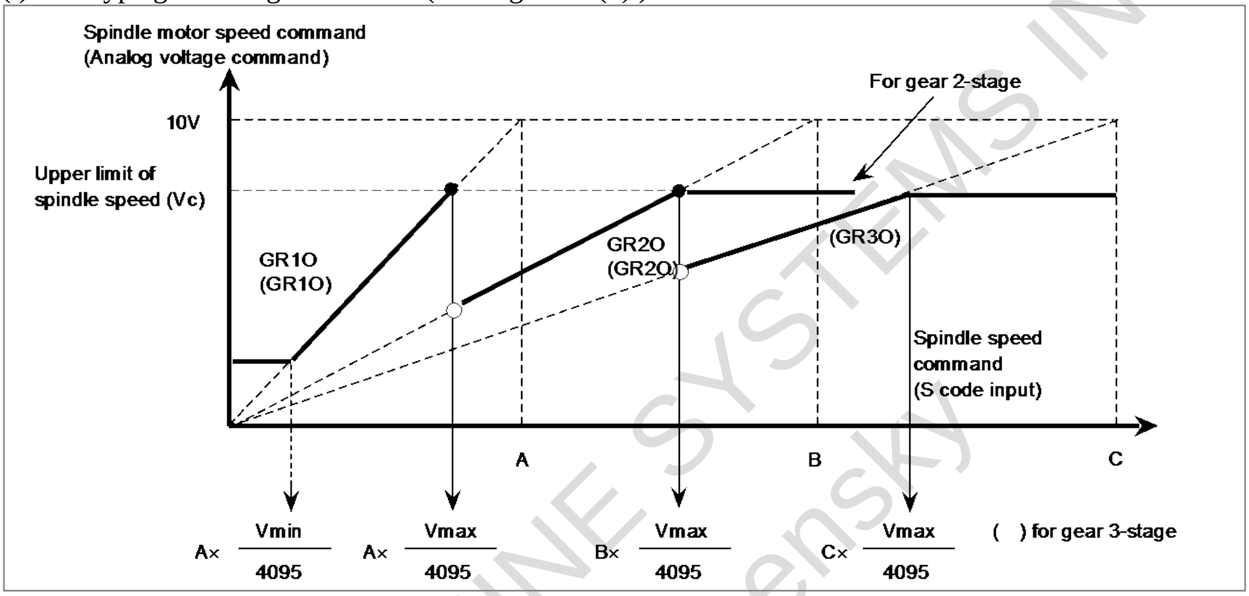


Fig. 10.5 (b) S code input and output voltage

Set the following values as parameters:

- Constant Vmax: for upper limit of spindle speed (parameter No. 3736)

$$\text{Vmax} = 4095 \times \frac{\text{Upper limit of spindle speed}}{\text{Spindle speed at command voltage 10V}}$$

- Constant Vmin; for lower limit of spindle speed (parameter No. 3735)

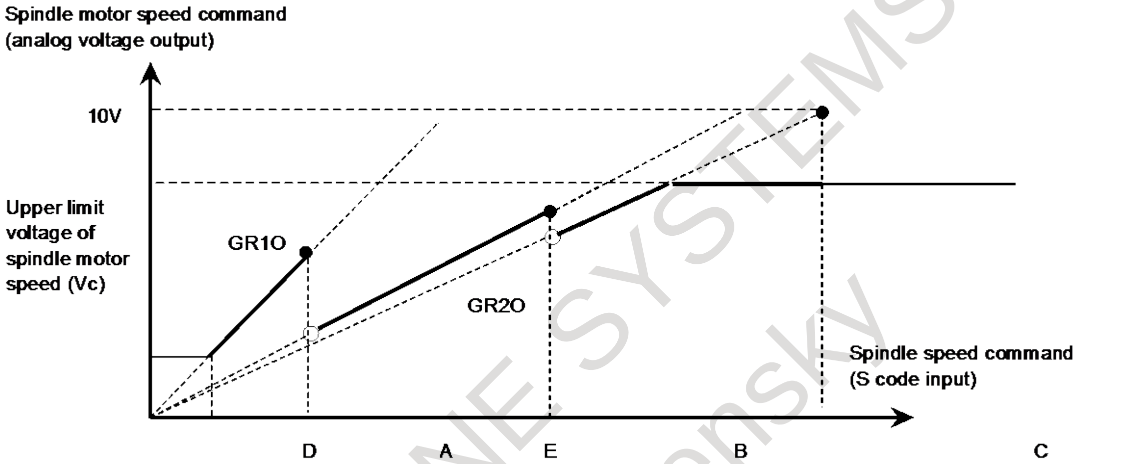
$$\text{Vmin} = 4095 \times \frac{\text{Lower limit of spindle speed}}{\text{Spindle speed at command voltage 10V}}$$

- Spindle speed A (min^{-1}) ; at command voltage 10V and low gear (parameter no. 3741)
- Spindle speed B (min^{-1}) ; at command voltage 10V and high gear (or middle-high gear) (parameter no. 3742)
- Spindle speed C (min^{-1}) ; at command voltage 10V and high gear (parameter no. 3743)(Gear 3-stage)

NOTE

If a specified voltage of 10 V is already higher than the acceptable input voltage for the spindle drive system, calculate hypothetically the spindle speed that corresponds to 10 V using a proportional calculation method and use it instead. Now, in response to the specified S code, the speed command and gear select commands (GR3O, GR2O, GR1O) are output to the spindle motor as shown in Fig. 10.5 (b).

(ii) Gear change point during tapping cycle mode (G84, G74)
In case of G84 (tapping cycle) or G74 (counter tapping cycle) the gear shift speed is changed by bit 3 (SGT) of parameter No. 3705. In this case, gear shift is performed at the speed set by parameters Nos. 3761 and 3762 (Fig. 10.5 (c)).



D : Gear shift point (parameter No. 3761) for low/high speed change (or low/medium speed change) (set by parameter as min^{-1} value) during tapping.
E : Gear shift point (parameter No. 3762) (3-stage gear) for medium/high speed change (set by parameter as min^{-1} value) during tapping.

Fig. 10.5 (c) S code input and output voltage (in tapping)

M

(iii) M type gear change method B (M series) (See Fig. 10.5 (d).)
The speed (min^{-1}) at which the low-speed and the high-speed gears are changed can be set as parameters Nos. 3751 and 3752 by setting bit 2 (SGB) of parameter No. 3705. When a 3-stage gear is used, it is possible to set the speeds (min^{-1}) for switching low-speed and medium-speed gears, and medium-speed and high-speed gears, using parameters Nos. 3751 and 3752.

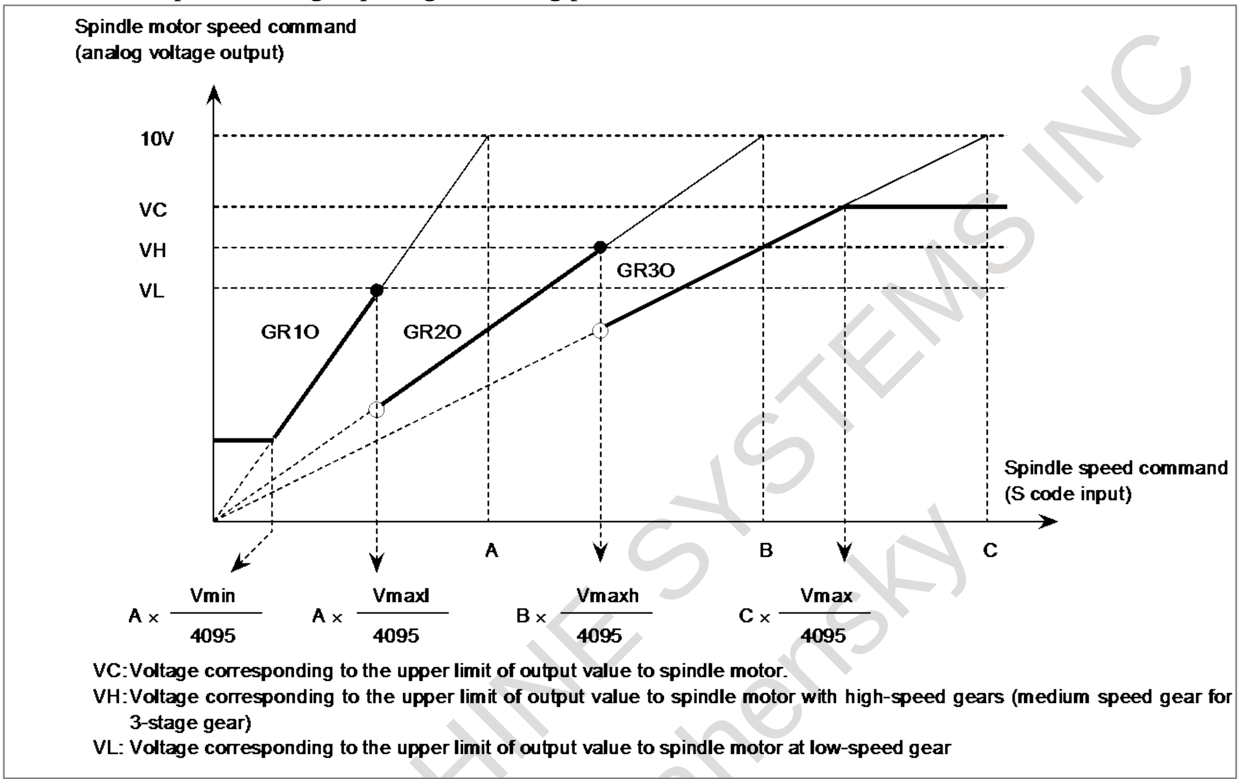


Fig. 10.5 (d) M type gear change B

When using this function, set the following parameters:

- Constant Vmax (Parameter No. 3736) related to the upper limit of spindle motor speed (min^{-1})

$$\text{Vmax} = 4095 \times \frac{\text{Upper limit of spindle motor speed}}{\text{Spindle motor speed when the command voltage is 10V}}$$

- Constant Vmin (Parameter No. 3735) related to the lower limit of spindle motor speed (min^{-1})

$$\text{Vmin} = 4095 \times \frac{\text{Lower limit of spindle motor speed}}{\text{Spindle motor speed when the command voltage is 10V}}$$

- Constant Vmaxl (Parameter No. 3751) related to the upper limit of spindle motor speed (min^{-1}) with low-speed gears

$$\text{Vmaxl} = 4095 \times \frac{\text{Upper limit of spindle motor speed with low-speed gears}}{\text{Spindle motor speed when the command voltage is 10V}}$$

- Constant Vmaxh (Parameter No. 3752) related to the upper limit of spindle motor speed (min^{-1}) with high-speed gears (medium-speed gear for 3-stage gear)

$$\text{Vmaxh} = 4095 \times \frac{\text{Upper limit of spindle motor speed with high-speed gears}}{\text{Spindle motor speed when the command voltage is 10V}}$$

- Spindle speed A (Parameter No. 3741) (min^{-1}) with low-speed gears when the command voltage is 10V
- Spindle speed B (Parameter No. 3742) (min^{-1}) with high-speed gears when the command voltage is 10V (medium-speed gear for 3-stage)
- Spindle speed C (Parameter No. 3743) (min^{-1}) with high-speed gears when the command voltage is 10V (3-stage gear)

Spindle motor speed commands and gear selecting signals (GR1O, GR2O, GR3O) are issued on each S code command as shown in the Fig. 10.5 (d):

> ⚠ **CAUTION**
> 1 In a tapping cycle when bit 3 (SGT) of parameter No. 3705 is set, the gears are changed over at the gear changing point for tapping.
> 2 For this function (bit 2 (SGB) of parameter No. 3705 = 1), when only one-stage gear is used, the voltage corresponding to the upper limit value to the spindle motor is calculated using Vmaxl, and when 2-stage gear is used, it is calculated according to Vmaxh. Therefore, when SGB is 1, set Vmaxl when only one-stage gear is used, Vmaxl and Vmaxh when 2-stage gear is used.

Bit 6 (GMB) of parameter No. 3712 can be used to select the maximum spindle motor speed for each gear.

When the bit 6 (GMB) of parameter No. 3712 is set to 0:

The maximum motor speed when each gear is used is clamped by parameters No. 3741 to 3743 or parameter No. 3736.

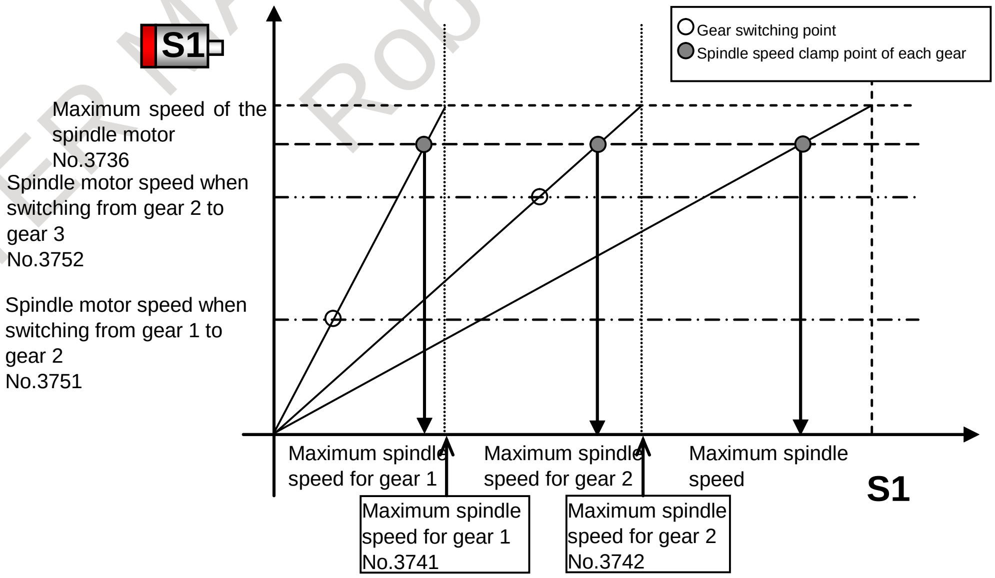


Fig. 10.5 (e) When the bit 6 (GMB) of parameter No. 3712 is set to 0

When the bit 6 (GMB) of parameter No. 3712 is set to 1:

The maximum motor speed of gear 1 is clamped at the motor speed of the switching point from gear 1 to gear 2 and the maximum motor speed of gear 2 is clamped at the motor speed of the switching point from gear 2 to gear 3.

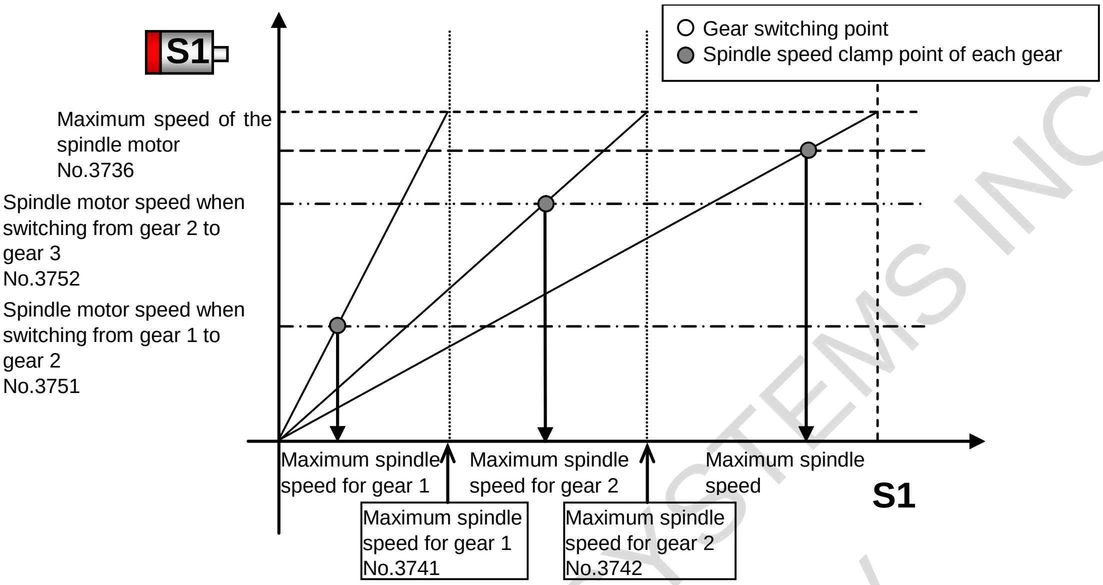
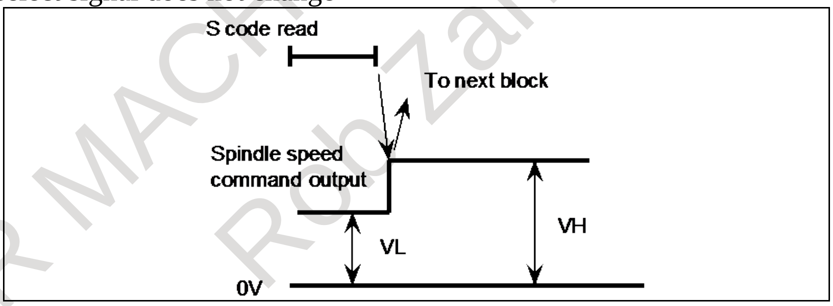


Fig. 10.5 (f) When the bit 6 (GMB) of parameter No. 3712 is set to 1

- Time chart

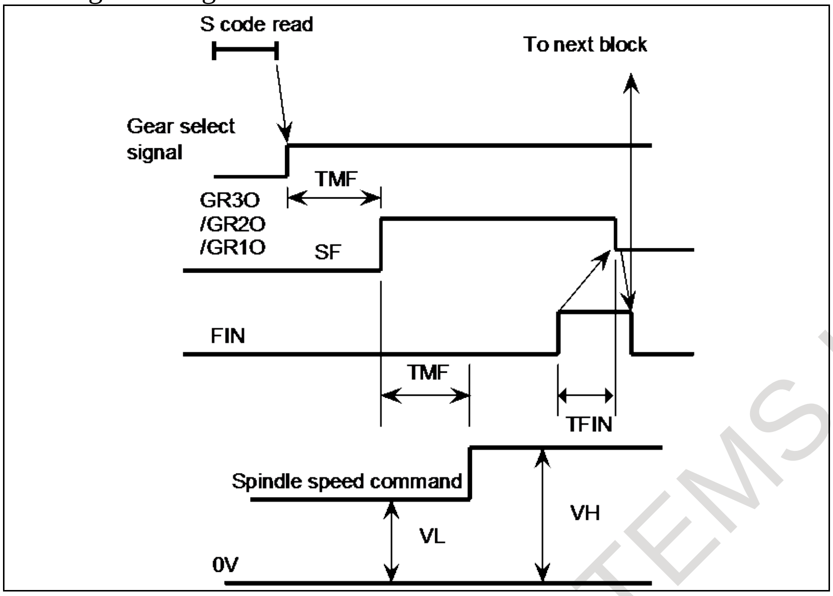
When S code is commanded, the I/O signal time chart is :

(i) When Gear select signal does not change



In this case, the SF signal is not output and the CNC advances to the next block automatically after the next spindle speed command is output.

(ii) When Gear select signal change



In this case, the gear select signal is output; after elapse of the time constant set by parameter (TMF), the SF signal is output. After another TMF elapse, the spindle speed command is output. On the PMC side, change the gears by this SF signal, and return the FIN signal after the end of gear change. The time chart for SF and FIN signals is the same as in S code output. TMF, set by parameter No. 3010, is common to M, S and T functions.
Moreover, specifying bit 6 (SFA) of parameter No. 3705 can specify that the SF signal be output even if no gear change is used.

- Details of T type (Input of GR1, GR2)

To perform the T type gear change, the maximum spindle speed for each gear side must be set in parameters Nos. 3741 to 3744.
The gear select signal is a 2 bit code signal (GR1, GR2). The relationship between the signal and gear number is Table 10.5 (b):

Table 10.5 (b)

Gear select signal		Gear No.	Parameter No. for max. spindle speed
GR1	GR2		
"0"	"0"	1	No. 3741
"1"	"0"	2	No. 3742
"0"	"1"	3	No. 3743
"1"	"1"	4	No. 3744

The following descriptions apply to the analog spindle. Like the descriptions of the M type, they also apply to the serial spindle on the assumption that spindle motor speed with analog voltage 10 V corresponds to the maximum spindle motor speed.
In addition, for the speed command output to the spindle motor, analog voltages 0 to 10 V for analog spindle control correspond to digital data 0 to 16383 for serial spindle control. However, it might be easier if you consider them code signals from 0 to 4095 for convenience sake without distinguishing between serial and analog spindles.
Assume that gear switching is two stage switching. If the spindle speed with the output voltage 10 V is 1000 min^{-1} for the low speed gear (G1) and 2000 min^{-1} for the high speed gear (G2), set these speeds by the parameters Nos. 3741 and 3742. In this case, the analog voltage has the linear relationship shown below (Fig. 10.5 (g)).

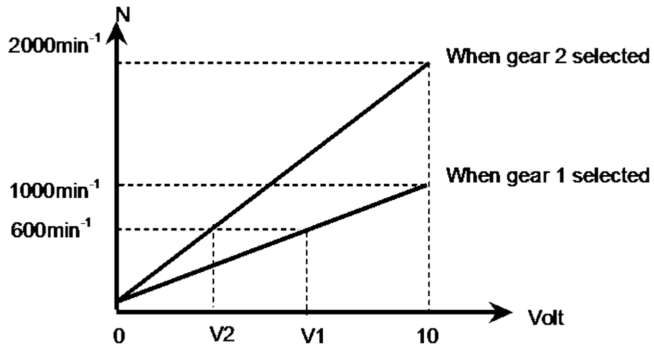


Fig. 10.5 (g)

When spindle speed S=600 is given, V1 (for G1) or V2 (for G2) in the Fig. 10.5 (g) is calculated inside the CNC and output to the machine side.
V1 ; 6 (V)
V2 ; 3 (V)
The value of output voltage V is calculated automatically from the following equations:
V = 10N / R
R : Spindle speed at 10V output voltage
N : Spindle speed given by S command of 5 digits
This is equivalent to the G97 mode for constant surface speed control.
See "CONSTATNT SURFACE SPEED CONTROL" for operations during the constant surface speed control mode (G96).

- Determining spindle speed command output R01O-R12O (Output)

Using the above processing for gear change, the CNC calculates the speed command output to the spindle motor that is necessary to obtain the specified spindle speed with the gear.
The calculation result is output to the PMC through S12-bit code signals indicating a value 0 to 4095. (R12O to R01O<Fn037.3 to Fn036.0>)
Then, according to the selection by the signal used to select the spindle motor speed command SIND <Gn033.7>, the speed command output to the spindle motor is determined.

SIND signal <Gn033.7>	Speed command output to spindle motor
"0"	Speed command output calculated by CNC
"1"	Speed command output set by PMC Also see "SPINDLE OUTPUT CONTROL BY THE PMC".

- Polarity of spindle speed command output

The speed command output to the spindle motor is determined as described above, but the actual output polarity is determined by the CNC as follows (Table 10.5 (c)):

Table 10.5 (c)

Bit 7 (TCW) of parameter No. 3706	Bit 6 (CWM) of parameter No. 3706	Output polarity
0	0	+(M03/M04 command)
0	1	-(M03/M04 command)
1	0	+(M03 command), -(M04 command)
1	1	-(M03 command), +(M04 command)

Then, according to the selection by the signal used to select the polarity of the spindle motor speed command SSIN <Gn033.6>, the output polarity of the speed command output to the spindle motor is determined.

SSIN signal <Gn033.6>	Output polarity of speed command output to spindle motor
"0"	Output polarity determined by CNC
"1"	Output polarity set by PMC Also see "SPINDLE OUTPUT CONTROL BY THE PMC".

NOTE

When bit 7 (TCW) of parameter No. 3706 is 1, and the output polarity is to be determined by the CNC, the CNC cannot determine the output polarity if M03 or M04 has not been specified even once since power-on. As a result, no command is output to the spindle.

- Command output to the spindle control unit

According to the speed command output and the polarity determined so far, the command is sent to the spindle control unit as follows:

- For serial spindle → Digital data 0 to ±16383
- For analog spindle → Analog voltage 0 to ±10 V

- Conditions for stopping spindle speed command output

A command being output to the spindle is set to 0 when *SSTP is set to “0” and when a command that sets the spindle speed command output to 0 such as a S0 command is specified.
With M05, emergency stop, and a reset, the CNC does not set the command output to the spindle to 0.

- Position coder feedback signal

The position coder is necessary for thread cutting or feed per revolution.
The position coder detects the actual spindle speed and the one-rotation signal (used to detect a fixed point on the spindle for threading). Number of pulses output from the position coder is set in parameter No. 3720.
When the gear ratio between the position coder and spindle is to be used, set the gear ratio for the position coder and spindle in parameters Nos. 3721 and 3722, respectively.
However, in the machine to do thread cutting, connect mechanically between the spindle and position coder by the gear ratio=1:1. And these parameters set the No.3721=0, No.3722=0. (It is the setting for the gear ratio 1:1.)
In the general thread cutting, the same thread grooves must be cut several times. To cutting the same thread groove repeatedly, it is necessary to detect the fixed point on the spindle. (One fixed point in the one rotation of the spindle.) One-rotation signal is used for the detection of the one fixed point. Typically, the one-rotation signal is generated on the one fixed point in the one rotation of the position coder.
For these reasons, when doing the thread cutting, the position coder and spindle must be connected by the gear ratio 1:1.
See "RIGID TAPPING" for position coder connection for rigid tapping.

- Speed arrival signal SAR <Gn029.4>

The spindle speed arrival signal SAR is an input signal used as a condition for the CNC to start cutting feed. This signal is used generally when cutting feed should be started after the spindle reaches the specified speed.
In this case, a sensor is used to check the spindle speed. The detected speed is sent to the CNC via the PMC.
When the above operation is performed continuously using the PMC ladder, however, cutting feed may be started based on the SAR signal indicating the previous spindle state (spindle speed before change), if the spindle speed change command and the cutting feed command are issued at the same time.
To avoid the above problem, monitoring the SAR signal can be deferred for a time specified by parameter No. 3740 after the S command or cutting feed command was issued.
When using the SAR signal, set bit 0 (SAR) of parameter No. 3708 to 1.

Item No. 0000 (SPEED ARRIVAL ON) on the diagnosis screen is kept at 1 while this function is keeping the cutting feed block at a halt.

- Reading actual spindle speeds by PMC window

The actual speed of the serial spindles can be read by PMC window (function code 138). Actual spindle speed is calculated by multiplying the ratio of the gear and motor speed. The gear which is in use is distinguished by the following signals.

Table 10.5 (d) Gear selection status

<table>
<tr><th rowspan="4">Gear number</th><th colspan="7">Bit 1 (SLG) of parameter No.3787</th></tr>
<tr><th colspan="2">0</th><th colspan="5">1</th></tr>
<tr><th colspan="2" rowspan="2">Clutch/gear signals (serial spindle)</th><th colspan="3">M type gear selection method</th><th colspan="2">T type gear selection method</th></tr>
<tr><th colspan="3">Gear selection signals</th><th colspan="2">Gear selection signals</th></tr>
<tr><th></th><th>CTH1</th><th>CTH2</th><th>GR3O</th><th>GR2O</th><th>GR1O</th><th>GR2</th><th>GR1</th></tr>
<tr><td>1</td><td>"0"</td><td>"0"</td><td>"0"</td><td>"0"</td><td>"1"</td><td>"0"</td><td>"0"</td></tr>
<tr><td>2</td><td>"0"</td><td>"1"</td><td>"0"</td><td>"1"</td><td>"0"</td><td>"0"</td><td>"1"</td></tr>
<tr><td>3</td><td>"1"</td><td>"0"</td><td>"1"</td><td>"0"</td><td>"0"</td><td>"1"</td><td>"0"</td></tr>
<tr><td>4</td><td>"1"</td><td>"1"</td><td>-</td><td>-</td><td>-</td><td>"1"</td><td>"1"</td></tr>
</table>

- Analog spindle

Gain and offset

The analog spindle may require gain and offset voltage adjustment depending on the spindle motor speed amplifier being used.
The following parameters are available for such adjustment.

Gain adjustment data: Parameter No. 3730
Offset voltage compensation: Parameter No. 3731

Spindle enable signal ENB<Fn001.4>

Another output related to spindle control is the spindle enable signal ENB.
The ENB signal is logical “1” when a nonzero command output is sent to the spindle. If the command is logical 0, the ENB signal becomes logical “0”.
When the analog spindle is being used, an offset voltage in the spindle motor speed amplifier may cause the spindle motor to rotate at low speed even if the command output (in this case, analog voltage) to the spindle is zero. The ENB signal can be used to stop the motor in such a case.

Electrical specification of analog spindle interface

The ENB1/2 signal on the analog spindle interface is an interface signal that is turned on and off under the same conditions as of the spindle enable signal <Fn001.4>, and it can be used even when a serial spindle is used.
For related signals on the analog spindle interface, refer to the description of the analog spindle interface in the section of the spindle connection in "Connection Manual (Hardware)".

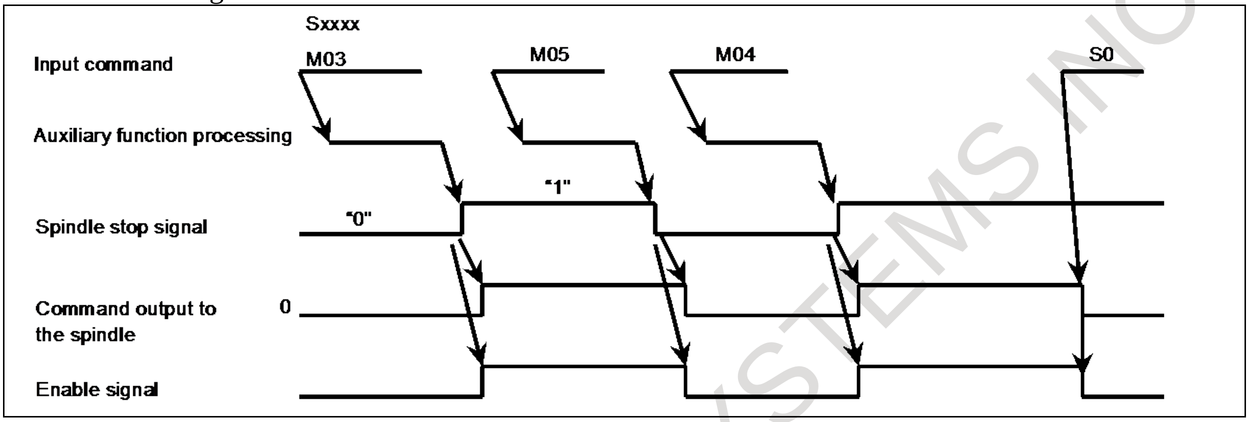
Signal

Spindle stop signal *SSTP<Gn029.6>

[Classification] Input signal

[Function] The command output to the spindle is disabled.

[Operation] When the spindle stop signal turns to “0”, the output voltage becomes 0V and the enable signal ENB turns to “0” (M05 is not output).
When this signal turns to “1”, the analog voltage returns to its original value and the ENB signal turns to “1”.



The above time chart is an example. Actually, the time chart should meet the specification of the spindle control unit.

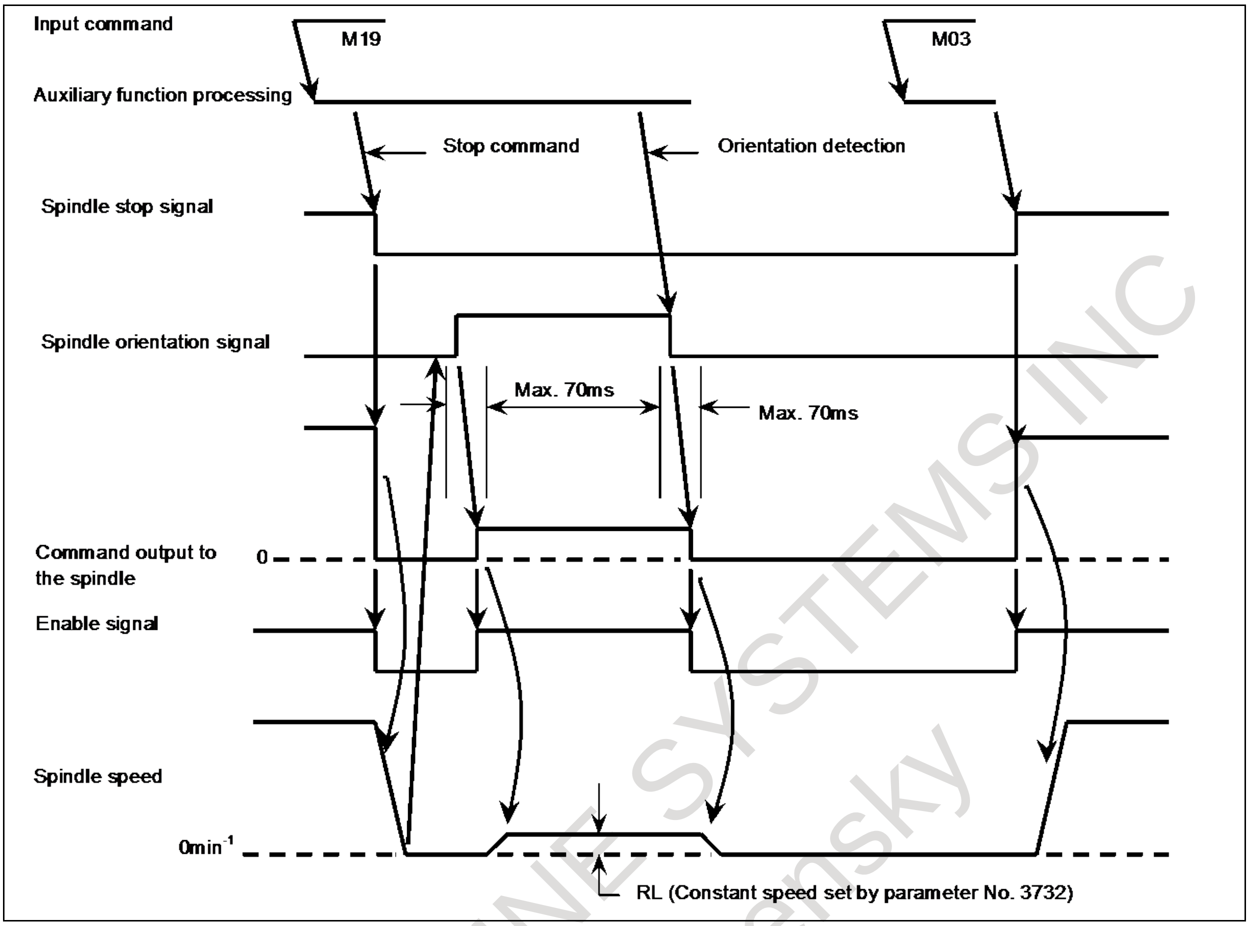
- When this signal is not used, always set the signal to “1”.
- When the M03, M04, or M05 command is issued, the CNC outputs the code signal and strobe signal only.

Spindle orientation signal SOR<Gn029.5>

[Classification] Input signal

[Function] The spindle or the spindle motor is rotated at a constant

[Operation] When the spindle orientation signal turns to “1” and the spindle stop signal *SSTP turns to “0”, a spindle speed command which lets the spindle rotate at the constant speed set by parameter No. 3732 is output. The enable signal ENB also turns to “1”. This signal is disabled when the spindle stop signal is “1”.
In the machining center system, when the spindle speed for orientation is set by bit 1 (GST) of parameter No. 3705 and the SOR signal is input, the CNC outputs the spindle speed command corresponding to the speed set to parameter 3732 with an output polarity set by bit 5 (ORM) of parameter No. 3706, but the gear select signal does not change.
For example, if the SOR signal is turned to 1 with high gear selected, and the speed set to parameter No. 3732 is in the low gear range, the gear select signal does not change and the command output is calculated and output to obtain the set speed at high gear.
When the spindle motor speed is set by bit 1 (GST) of parameter No. 3705=1, the command output is output regardless of gear select signal. When the spindle motor speed is set, it is used for gear shift.
Example of usage is shown below:



Spindle speed override signals SOV0 to SOV7<Gn030>

[Classification] Input signal

[Function] The spindle speed override signal specifies an override from 0% to 254% in 1% units for the S command sent to the CNC.

[Operation] An override value in binary must be set in 8 bits from SOV7 to SOV0.
The spindle speed override function is disabled (an override of 100% is applied) under the following conditions:

- Tapping cycle (M series : G84, G74 T series : G84, G88)
- Thread cutting (M series : G33 T series : G32, G92, G76)

→ When this function is not in use, specify an override of 100%; otherwise, an override of 0% becomes effective, thus disabling the spindle from rotating.

Spindle speed arrival signal SAR<Gn029.4>

[Classification] Input signal

[Function] The SAR signal initiates cutting feed. In other words, if the signal is logical "0", cutting feed will not start.

[Operation] Generally, this signal is used to inform the CNC that the spindle has reached the specified speed.
For this purpose, the signal must be set to "1" only after the actual speed of the spindle has reached the specified speed.
Setting parameter No. 3740 with a wait time before the start of checking the SAR signal inhibits cutting feed from starting under a condition of SAR = "1" specified before the change of the spindle command.
To use the SAR signal, it is necessary to set bit 0 (SAR) of parameter No. 3708 to 1.
The CNC checks the SAR signal under the following conditions:

a. Bit 0 (SAR) of parameter No. 3708 is set to 1.

b. Before starting distribution of the first feed (move command) block after shifting from the rapid traverse mode to the cutting feed mode. This checking is performed after the time set by parameter No. 3740 has elapsed after the feed block is read.

c. Before starting distribution of the first feed command block after an S code is commanded. The wait time for checking is the same as in item b.

d. When an S code and feed are programmed in the same block, the S code (or command output to the spindle) is output, and the SAR signal is checked after a fixed time elapses. If the SAR signal is set to “1”, feed begins.

> ⚠ **CAUTION**
> According to the conditions of item d above, note that if the circuit is so designed that SAR is turned to “0” simultaneously with the output of an S code and the change of spindle speed is initiated by the DEN signal, the operation will stop. That is, the spindle speed does not reach the commanded speed because the CNC is waiting for the DEN signal and distribution is not started because the CNC is waiting for the SAR signal.

Spindle enable signal ENB<Fn001.4>

[Classification] Output signal

[Function] Informs absence or presence of spindle output command.

[Output cond.] The ENB signal becomes logical “0” when the command output to the spindle becomes logical 0. Otherwise, the signal is logical “1”.
During analog spindle control, S0 may not be able to stop the spindle from rotating at low speed because of an offset voltage in the spindle motor speed control amplifier. In such a case, the ENB signal can be used to provide a condition to determine whether to stop the motor.
The analog spindle interface has electric signals (ENB1 and ENB2) similar to the ENB. These signals work under the same conditions as with the ENB signal.
The ENB signal can be used also for serial spindle control.

Gear selection signals GR1O, GR2O, GR3O<Fn034.0 to .2>

[Classification] Output signal

[Function] The gear select signal specifies a gear stage to the PMC.

[Output cond.] For details of this signal, see descriptions on the M type gear selection method in "Overview."

Gear selection signals GR1, GR2<Gn028.1, .2>

[Classification] Input signal

[Function] This signal informs the CNC of the gear stage currently selected.

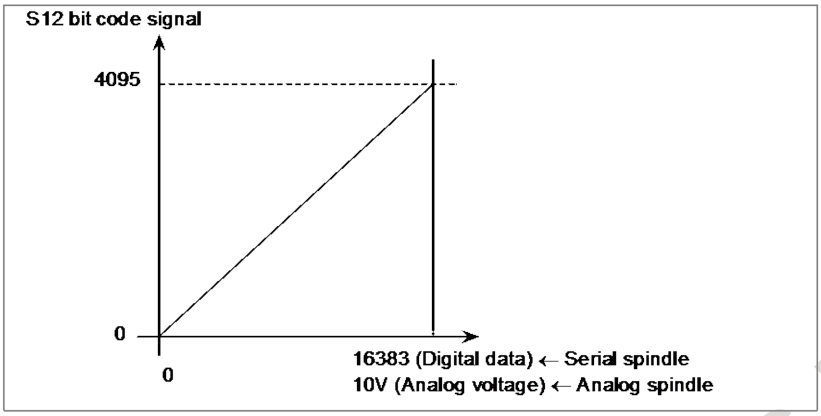
[Operation] For details of this signal, see descriptions on the T type gear selection method in "Overview."

S12-bit code signals R01O to R12O<Fn036.0 to Fn037.3>

[Classification] Output signal

[Function] These signals convert the spindle speed command value calculated by the CNC to code signals 0 to 4095.

[Output cond.] The relationship between the spindle speed command value (calculated by the CNC) and the value output by this signal is as shown below.



These signals convert the spindle speed command value calculated by the spindle control function of the CNC to data from 0 to 4095 (for both serial and analog spindle control) and outputs the result. This does not match the command set by spindle output control of the PMC. (See "SPINDLE OUTPUT CONTROL BY THE PMC".)

- Other signals

Spindle speed function code signals S00 to S31<Fn022 to Fn025> (Output)
Spindle speed function strobe signal SF<Fn007.2> (Output)

See "SPINDLE SPEED FUNCTIONS" and "SPINDLE OUTPUT CONTROL BY THE PMC" for these signals.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn028						GR2	GR1	
Gn029		*SSTP	SOR	SAR				
Gn030	SOV7	SOV 6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0
Fn001				ENB				
Fn007						SF		
Fn022	S07	S06	S05	S04	S03	S02	S01	S00
Fn023	S15	S14	S13	S12	S11	S10	S09	S08
Fn024	S23	S22	S21	S20	S19	S18	S17	S16
Fn025	S31	S30	S29	S28	S27	S26	S25	S24
Fn034						GR3O	GR2O	GR1O
Fn036	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
Fn037					R12O	R11O	R10O	R09O

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				ESF
		SFA	NSF		SGT	SGB	GST	ESF

[Input type] Parameter input
[Data type] Bit path

#0 ESF When the spindle control function (Spindle analog output or Spindle serial output) is used, and the constant surface speed control function is used or bit 4 (GTT) of parameter No. 3706 is set to 1:
- 0: S codes and SF are output for all S commands.
- 1: For the T series:
  S codes and SF are not output for an S command in the constant surface speed control (G96) mode and a command for maximum spindle speed clamping (G92 S_; (G50 for G code system A)).
  For the M series:
  S codes and SF are not output for an S command in the constant surface speed control (G96) mode.

> **NOTE**
> The operation of this parameter varies between the T series and M series.
> For the T series:
> This parameter is valid when bit 4 (EVS) of parameter No. 3705 is set to 1.
> For the M series:
> For an S command for maximum spindle speed clamping (G92 S_;), SF is not output, regardless of the setting of this parameter.

#1 GST The SOR signal is used for:
- 0: Spindle orientation
- 1: Gear shift

#2 SGB Gear switching method is:
- 0: Method A (Parameters Nos. 3741 to 3743 for the maximum spindle speed at each gear are used for gear selection.)
- 1: Method B (When bit 2 (MCH) of parameter No. 3790 is set to 0, parameters Nos. 3751 to 3752 are used for gear selection. When bit 2 (MCH) of parameter No. 3790 is set to 1, parameters Nos. 3753 to 3754 are used for gear selection).

#3 SGT Gear switching method during tapping cycle (G84 and G74) is:
- 0: Method A (Same as the normal gear switching method)
- 1: Method B (Gears are switched during tapping cycle according to the spindle speed set in parameters Nos. 3761 and 3762).

#4 EVS When the spindle control function (Spindle analog output or Spindle serial output) is used, S codes and SF are:
- 0: Not output for an S command.
- 1: Output for an S command.

The output of S codes and SF for an S command in constant surface speed control mode (G96), or for an S command used to specify maximum spindle speed clamping (G92 S_; (G50 for G code system A)) depends on the setting of bit 0 (ESF) of parameter No. 3705.

#5 NSF For the M series, when a T type gear is selected (with bit 4 (GTT) of parameter No. 3706 set to 1 or constant surface speed control is enabled (bit 0 (SSC) of parameter No. 8133 is 1)), and an S code is specified:
- 0: SF is output.
- 1: SF is not output.

> **NOTE**
> This parameter does not affect S code output. For an S command for maximum spindle speed clamping (G92 S_;), SF is not output, regardless of the setting of this parameter.

#6 SFA The SF signal is output:
0: When gears are switched.
1: Irrespective of whether gears are switched.

	#7	#6	#5	#4	#3	#2	#1	#0
3706	TCW	CWM	ORM					
	TCW	CWM	ORM	GTT				

[Input type] Parameter input
[Data type] Bit path

#4 GTT Spindle gear selection method is:
0: Type M.
1: Type T.

> **NOTE**
> 1 M type
> The gear selection signal is not input. The CNC selects a gear based on the speed range of each gear set by a parameter beforehand according to S codes, and the selected gear is posted by outputting the gear selection signal. Moreover, the spindle speed matching the gear selected by the output gear selection signal is output.
> T type
> The gear selection signal is input. The spindle speed matching the gear selected by this signal is output.
> 2 When the constant surface speed control is enabled (bit 0 (SSC) of parameter No. 8133 is 1), type T is selected, regardless of whether this parameter is specified.
> 3 When type T spindle gear switching is selected, the following parameters have no effect:
> No.3705#2(SGB), No.3751, No.3752,
> No.3705#1(GST), No.3705#3(SGT), No.3761, No.3762,
> No.3705#6(SFA), No.3735, No.3736
> On the other hand, parameter No. 3744 becomes usable.

#5 ORM Voltage polarity during spindle orientation
0: Positive
1: Negative

#6 CWM
#7 TCW Voltage polarity when the spindle speed voltage is output

TCW	CWM	Voltage polarity
0	0	Both M03 and M04 positive
0	1	Both M03 and M04 negative
1	0	M03 positive, M04 negative
1	1	M03 negative, M04 positive

	#7	#6	#5	#4	#3	#2	#1	#0
3708		TSO					SAT	SAR
		TSO						SAR

[Input type] Parameter input
[Data type] Bit path

#0 SAR The spindle speed arrival signal SAR is:
0: Not checked
1: Checked

#1 SAT Check of the spindle speed arrival signal SAR at the start of executing the thread cutting block
0: The signal is checked only when bit 0 (SAR) of parameter No. 3708 is set to 1.
1: The signal is always checked irrespective of the setting of SAR.

NOTE
When thread cutting blocks are consecutive, the spindle speed arrival signal is not checked for the second and subsequent thread cutting blocks.

#6 TSO During a threading or tapping cycle, the spindle override is:
0: Disabled (tied to 100%).
1: Enabled.

NOTE
During rigid tapping, the override is tied to 100%, irrespective of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3709								SAM

[Input type] Parameter input
[Data type] Bit path

#0 SAM The sampling frequency to obtain the average spindle speed
0: 4 (Normally, set to 0.)
1: 1

	#7	#6	#5	#4	#3	#2	#1	#0
3712		GMB						

[Input type] Parameter input
[Data type] Bit

#6 GMB With type-M gear switching method B, the speed of each gear is clamped to:
0 : The maximum rotation speed (No. 3741 to No. 3743) of each gear or the maximum clamping speed (No. 3736) of the spindle motor.
1 : Each gear switching point by parameters Nos. 3751 to 3752 in case of bit 2 (MCH) of parameter No. 3790 is set to 0. Each gear switching point by parameters Nos. 3753 to 3754 in case of bit 2 (MCH) of parameter No. 3790 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
3715								NSAx

[Input type] Parameter input
[Data type] Bit axis

#0 NSAx When a move command is executed for an axis, the spindle speed arrival signal SAR is:
0: Checked.
1: Not checked.
Set an axis for which the spindle speed arrival signal SAR need not be checked when a move command is executed for the axis. When a move command is specified only for an axis with this parameter set to 1, the spindle speed arrival signal SAR is not checked.

	#7	#6	#5	#4	#3	#2	#1	#0
3716								A/Ss

[Input type] Parameter input
[Data type] Bit spindle

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 A/Ss Spindle motor type is :
0: Analog spindle.
1: Serial spindle.

NOTE
1 The 1 analog spindle at the maximum can be controlled in a system.
2 To use a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0.
3 When using an analog spindle, assign it at the end of the spindle configuration.

3717	Spindle amplifier number to each spindle

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled axes
Set a spindle amplifier number to be assigned to each spindle.
0: No spindle amplifier is connected.
1: Spindle motor connected to amplifier number 1 is used.
2: Spindle motor connected to amplifier number 2 is used.
to
n: Spindle motor connected to amplifier number n is used.

3718	Subscript for display of a serial spindle (main spindle) or analog spindle

3719	Subscript for display of a serial spindle (sub-spindle)

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 122

Set a subscript to be added to spindle speed display on a screen such as the position display screen.

3720	Number of position coder pulses

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Detection unit
[Valid data range] 1 to 32767

Set the number of position coder pulses.

3721	Number of gear teeth on the position coder side

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 9999

Set the number of gear teeth on the position coder side in speed control (such as feed per revolution).

3722	Number of gear teeth on the spindle side

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 9999

Set the number of gear teeth on the spindle side in speed control (such as feed per revolution).

⚠ CAUTION
1 Parameter No.3721 and No.3722 can not be used for thread cutting. If thread cutting was done by setting the any value on these parameters, the correct screw can't be cut.
In the machine to do thread cutting, connect mechanically between the spindle and position coder by the gear ratio=1:1. And these parameters set the No.3721=0, No.3722=0.
(It is the setting for the gear ratio 1:1.)
2 Parameter No.3721 and No.3722 can not be used for Cs contour control. If Cs contour control was done by setting the any value on these parameters, spindle can’t move as commanded. In the machine to do Cs contour control, connect mechanically between the spindle and position coder by the gear ratio = 1:1. And these parameters set the No.3721=0, No.3722=0. (It is the setting for the gear ratio 1:1.)

3730	Data used for adjusting the gain of the analog output of spindle speed

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 0.1%
[Valid data range] 700 to 1250

Set data used for adjusting the gain of the analog output of spindle speed.

[Adjustment method]

<1> Assign standard value 1000 to the parameter.
<2> Specify the spindle speed so that the analog output of the spindle speed is the maximum voltage (10 V).
<3> Measure the output voltage.
<4> Assign the value obtained by the following equation to parameter No. 3730.
Setting value = (10 (V) / Measured data (V)) $\times$ 1000
<5> After setting the parameter, specify the spindle speed so that the analog output of the spindle speed is the maximum voltage. Confirm that the output voltage is 10V.

> **NOTE**
> This parameter needs not to be set for serial spindles.

3731	Compensation value for the offset voltage of spindle speed analog output

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Velo
[Valid data range] -1024 to 1024

Set a compensation value for the offset voltage of spindle speed analog output.
Setting = -8191 $\times$ offset voltage (V)/12.5

[Adjustment method]

<1> Assign standard value 0 to the parameter.
<2> Specify the spindle speed so that the analog output of the spindle speed is 0.
<3> Measure the output voltage.
<4> Assign the value obtained by the following equation to parameter No. 3731.

$$\text{Setting value} = \frac{-8191 \times \text{Offset voltage (V)}}{12.5}$$

<5> After setting the parameter, specify the spindle speed so that the analog output of the spindle speed is 0. Confirm that the output voltage is 0V.

> **NOTE**
> This parameter needs not to be set for serial spindles.

3732	The spindle speed during spindle orientation or the spindle motor speed during spindle gear shift

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 20000

Set the spindle speed during spindle orientation or the spindle motor speed during gear shift.

When bit 1 (GST) of parameter No. 3705 is set to 0, set the spindle speed during spindle orientation in min^{-1}.

When bit 1 (GST) of parameter No. 3705 is set to 1, set the spindle motor speed during spindle gear shift calculated from the following formula.

For a serial spindle

$$\text{Setting value} = \frac{\text{Spindle motor speed during spindle gear shift}}{\text{Maximum spindle motor speed}} \times 16383$$

For an analog spindle

$$\text{Setting value} = \frac{\text{Spindle motor speed during spindle gear shift}}{\text{Maximum spindle motor speed}} \times 4095$$

3735	**Minimum clamp speed of the spindle motor**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 4095

Set the minimum clamp speed of the spindle motor.

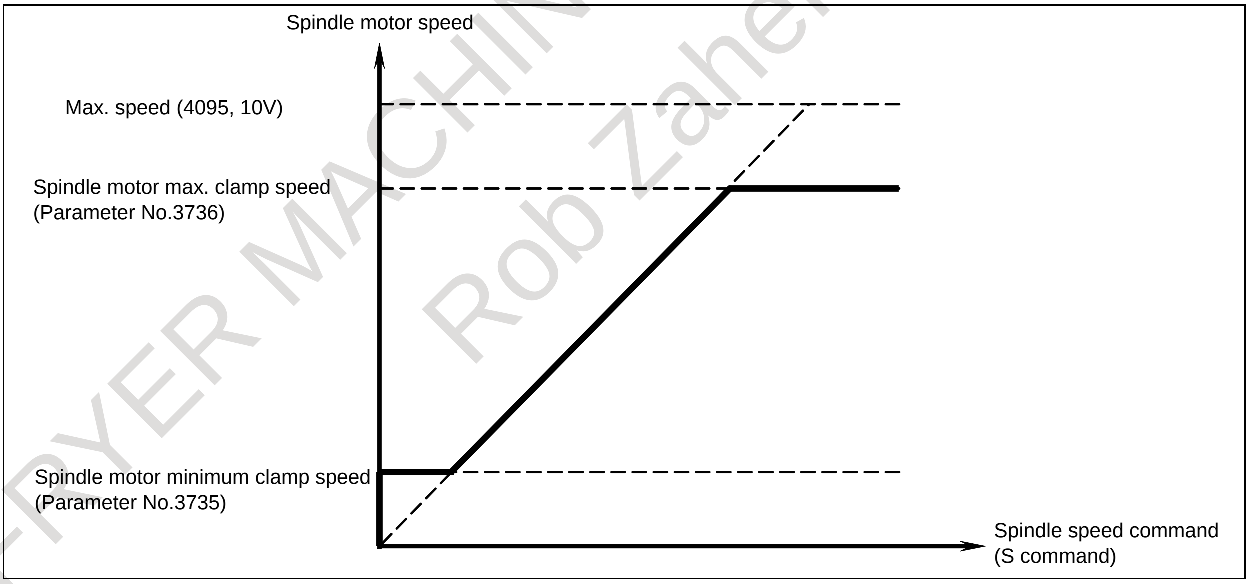
$$\text{Setting value} = \frac{\text{Minimum clamp speed of the spindle motor}}{\text{Maximum spindle motor speed}} \times 4095$$

3736	**Maximum clamp speed of the spindle motor**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 4095

Set the maximum clamp speed of the spindle motor.

$$\text{Setting value} = \frac{\text{Maximum clamp speed of the spindle motor}}{\text{Maximum spindle motor speed}} \times 4095$$



3740	**Time elapsed prior to checking the spindle speed arrival signal**

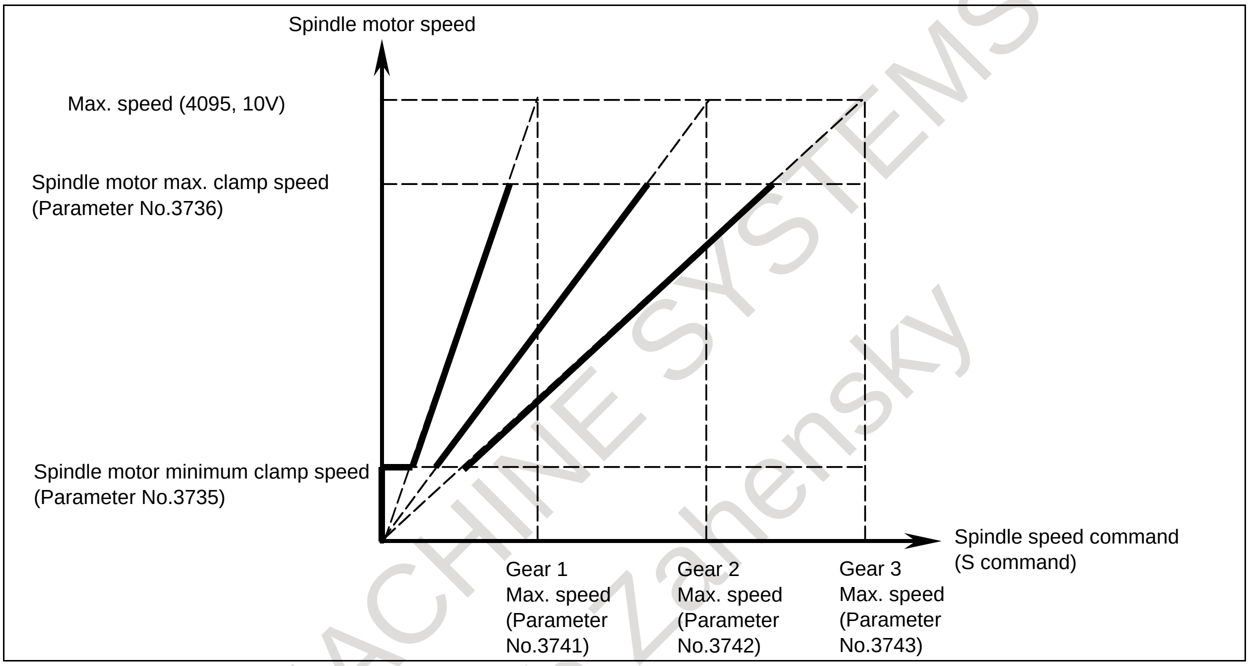
[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 32767

Set the time elapsed from the execution of the S function up to the checking of the spindle speed arrival signal.

3741	**Maximum spindle speed for gear 1**
3742	**Maximum spindle speed for gear 2**
3743	**Maximum spindle speed for gear 3**
3744	**Maximum spindle speed for gear 4**

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

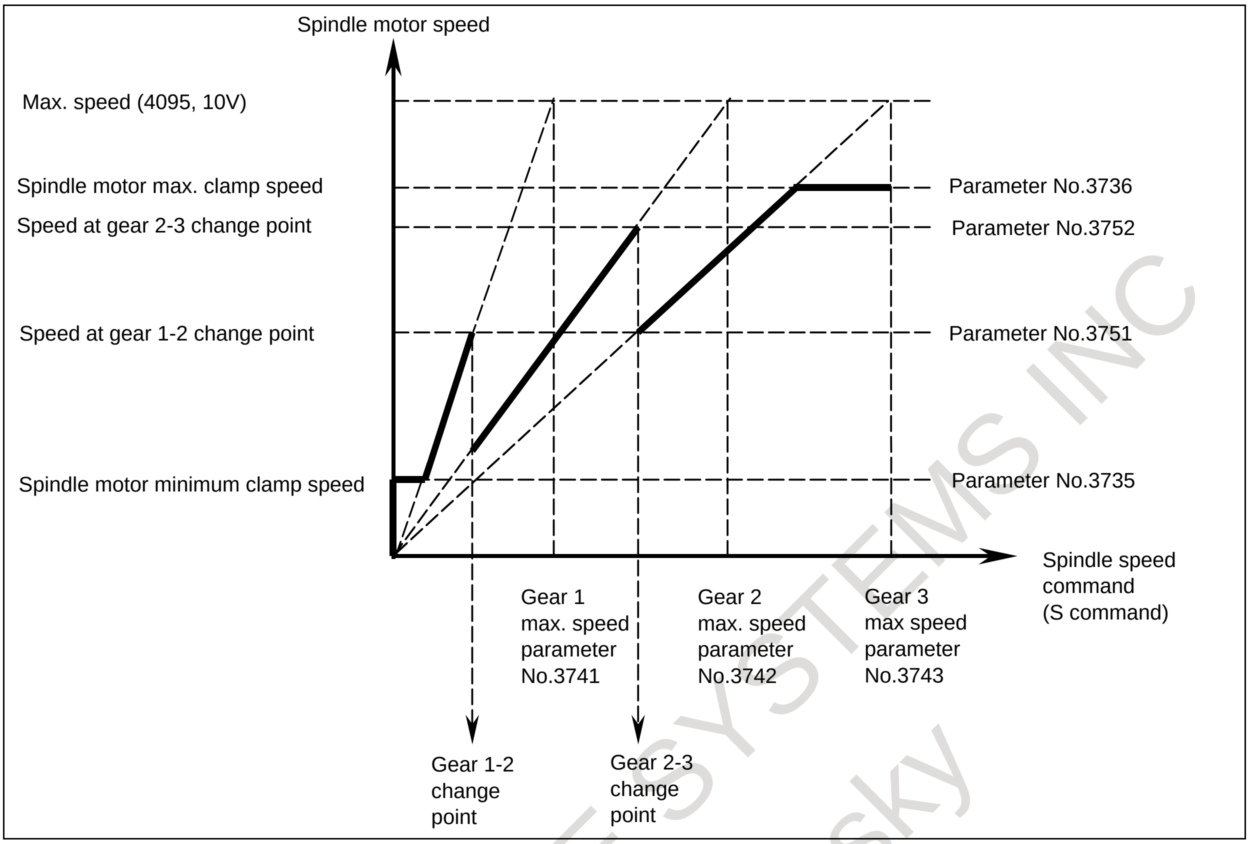
Set the maximum spindle speed corresponding to each gear.



3751	**Spindle motor speed when switching from gear 1 to gear 2**
3752	**Spindle motor speed when switching from gear 2 to gear 3**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 4095

For gear switching method B, set the spindle motor speed when the gears are switched.
Setting value =
(Spindle motor speed when the gears are switched / Maximum spindle motor speed) × 4095

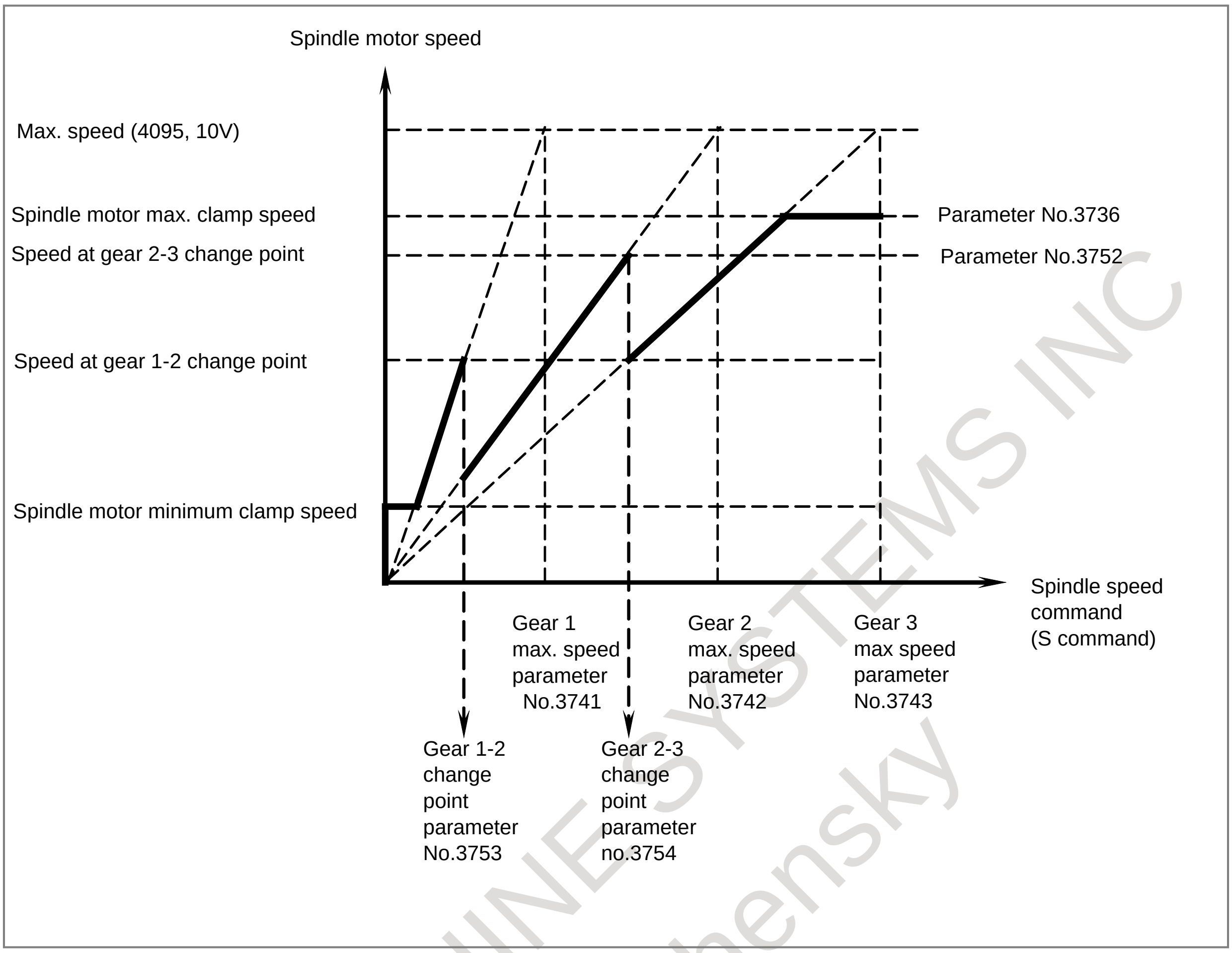


3753	
	Spindle speed when switching from gear 1 to gear 2

3754	
	Spindle speed when switching from gear 2 to gear 3

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

When method B is selected as the gear change method (when bit 2 (SGB) of parameter No. 3705 is set to 1) and bit 2 (MCH) of parameter No. 3790 is set to 1, each gear switching point can be set by the spindle speed.

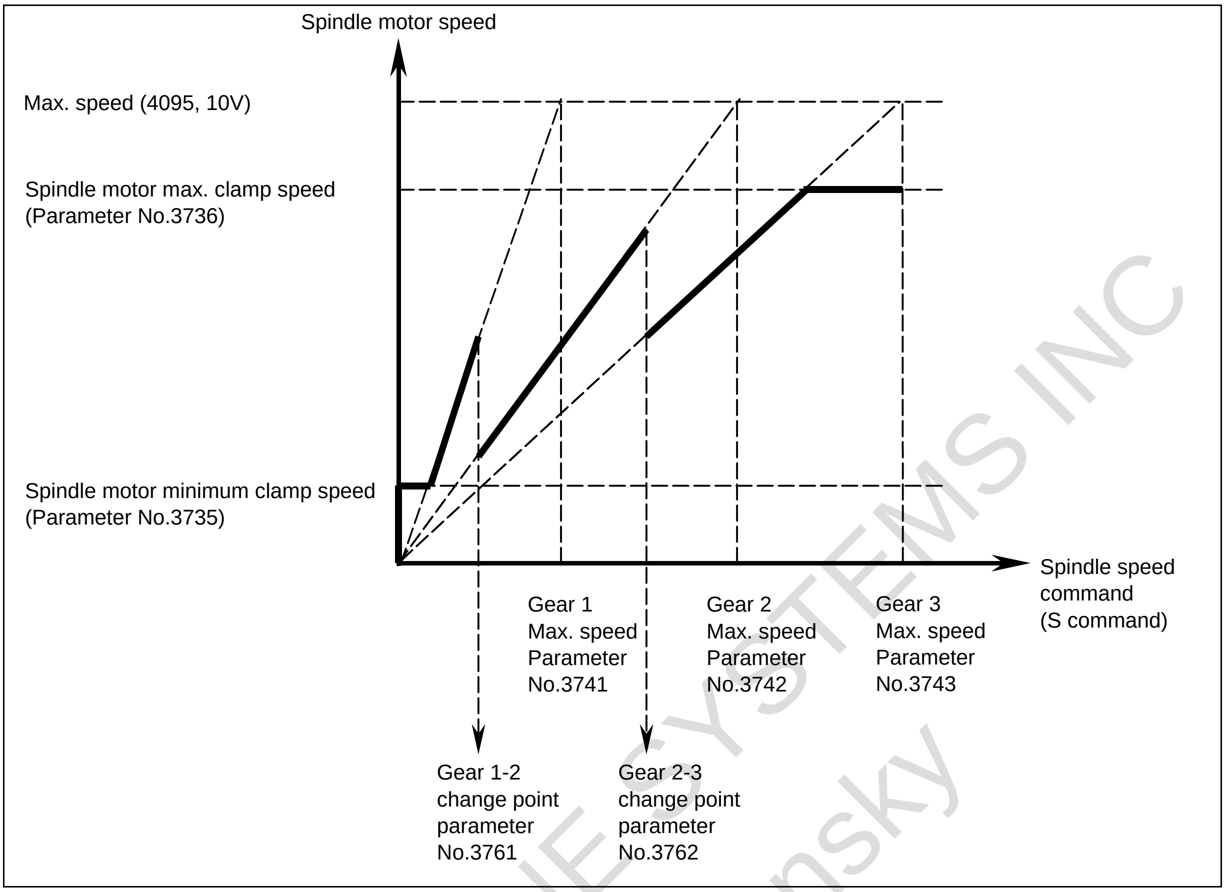


3761	Spindle speed when switching from gear 1 to gear 2 during tapping

3762	Spindle speed when switching from gear 2 to gear 3 during tapping

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

When method B is selected as the gear change method in the tapping cycle (when bit 3 (SGT) of parameter No. 3705 is set to 1), set the spindle speed at a change point of each gear.



3772	Maximum spindle speed

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

This parameter sets the maximum spindle speed.
When a command specifying a speed exceeding the maximum speed of the spindle is specified, or the speed of the spindle exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

> ⚠ **CAUTION**
> 1 When 0 is set in this parameter, the speed of the spindle is not clamped.
> 2 When spindle speed command control is applied using the PMC, this parameter has no effect, and the spindle speed is not clamped.

> **NOTE**
> 1 For M series, this parameter is valid if the function of constant surface speed control is enabled (bit 0 (SSC) of parameter No.8133 is 1).
> 2 When the constant surface speed control is enabled (bit 0 (SSC) of parameter No. 8133 is 1), the spindle speed is clamped at the maximum speed, regardless of whether the G96 mode or G97 mode is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
3787							SLG	

[Input type] Parameter input
[Data type] Bit spindle

#1 SLG In spindle speed calculation, gear selection status is judged from:
0 : Clutch/gear signals.
1 : Gear selection signals.

> **NOTE**
> 1 Address of clutch/gear signals is as follows.
> CTH1A, CTH2A<Gn070.3, Gn070.2> (First spindle)
> CTH1B, CTH2B<Gn074.3, Gn074.2> (Second spindle)
> CTH1C, CTH2C<Gn204.3, Gn204.2> (Third spindle)
> CTH1D, CTH2D<Gn266.3, Gn266.2> (Fourth spindle)
> 2 Address of gear selection signals is as follows.
> M type gear selection method
> GR1O, GR2O, GR3O<Fn034.0, Fn034.1, Fn034.2>
> T type gear selection method
> GR1, GR2 <Gn028.1, Gn028.2> (First spindle)
> GR21, GR22<Gn029.0, Gn029.1> (Second spindle)
> GR31, GR32<Gn029.2, Gn029.3> (Third spindle)
> GR41, GR42<Gn031.4, Gn031.5> (Fourth spindle)

	#7	#6	#5	#4	#3	#2	#1	#0
3790						MCH		KSA

[Input type] Parameter input
[Data type] Bit path

#0 KSA In the thread cutting (G33(M), G32,G34,G35,G36,G76,G92(T)) and polygon turning (G51.2(T)) using the analog spindle, the value of the speed indication of the spindle until the one rotation signal is detected at the start of thread cutting,
0: It may be temporarily low, possibly 0.
1: It does not change at the speed just before the start.

#2 MCH In spindle gear selection method type M,
0: It can be set from 0 to 4095 by parameter Nos. 3751 to 3752 with the maximum motor speed at the gear switching point as 4095.
1: The switching point of the gear can be set by spindle speed with parameter Nos. 3753 to 3754.

3792	The sampling frequency to obtain the average spindle speed

[Data type] Byte spindle
[Unit of data] No unit
[Valid data range] 0 to 4
The sampling frequency is 2^(parameter data).

NOTE
1. If this parameter is 0 or out of range, the sampling frequency to obtain the average spindle speed obeys SAM (parameter No.3709#0). If you would like to set sampling frequency to 1, set this parameter to 0, and SAM (parameter No.3709#0) to 1.
2. If you change this parameter, operate on the condition that spindle rotation is stop state and the function to use spindle feedback such as feed per revolution is not executed.

	#7	#6	#5	#4	#3	#2	#1	#0
3799		SPC						

[Input type] Parameter input
[Data type] Bit spindle

#6 SPC The position coder pulse to obtain the average spindle speed is:
0 : Sampled without sign data.
1 : Sampled with sign data.

Caution

⚠ **CAUTION**
This section mentioned a spindle speed control that should be prepared on the CNC side. But it is also necessary to design the processing of the signals to the spindle control unit.
Refer to the manual of the spindle control unit used and take necessary actions on the spindle control unit.

10.6 SPINDLE OUTPUT CONTROL BY THE PMC

Overview

The PMC can control the speed and polarity of each spindle motor, connected.
The first to fourth spindles each have their own individual interfaces. By using a PMC ladder program, the user can control the spindles as desired. This section describes how to use the PMC to control spindle rotation and provides example applications.

NOTE
Ladder sequence of the spindle output control by the PMC can be easily incorporated by using PMC function. The FANUC LADDER-III CD, function block that can be incorporated easily spindle output control by the PMC is stored in the form of a library.
For details on library of function block, refer to the documents in the FANUC LADDER-III CD. For details on function block, refer to the PMC Programming Manual (B-64513EN).

Explanation

- Switching control

This function can be used to specify the following:

- Spindle motor speed (number of rotations)
- Output polarity for each spindle motor (direction of rotation)

Usually, the CNC is used to control the speed and polarity of the first spindle motor. If the multi-spindle control is added, the CNC can also control the second to fourth spindle motors.
This function allows the user to select whether the CNC or PMC is used to control the speed and output polarity of the spindle motors.

- Specifying the spindle motor speed

The PMC can be used to specify the spindle motor speed upon executing the following:

- Switching the controller from the CNC to the PMC, by issuing a signal used to select the spindle motor speed command SINDx
- Setting the spindle motor speed data, calculated by the PMC, in spindle control signals R01Ix to R12Ix

When controlled by the PMC, the spindle motor speed is not affected by any signal (for example, the spindle speed override signal) or parameter settings (for example, the maximum speed clamp parameter) related to the spindle speed command of the CNC spindle control function.

→ If the multi-spindle control is added, however, the spindle stop signal *SSTPx <Gn027.3, 4, 5, Gn026.6> can be used to stop a PMC-controlled spindle.

The spindle motor speed data is obtained from the following expression. Its value can range from 0 to 4095:

$$\text{Spindle motor speed data} = \frac{\text{Spindle motor speed}}{\text{Maximum spindle motor speed (parameter No. 4020)}} \times 4095$$

Remark)

Usually, the spindle speed must be controlled. If a gear train is used to connect the spindle to the spindle motor, first obtain the maximum spindle speed at the maximum spindle motor speed.

$$\text{Spindle motor speed data} = \frac{\text{Spindle speed}}{\text{Maximum spindle speed}} \times 4095$$

By using this expression, the spindle motor speed data can easily be obtained.

- Specifying the output polarity for the spindle motor

The PMC can specify the spindle motor output polarity when the following are executed:

- Switching the controller from the CNC to the PMC, by issuing a signal used to select the polarity of the spindle motor speed command SSINx
- Specifying the output polarity to the signal used to specify the polarity of the spindle motor selected by the PMC SGNx

- S-code and SF signals

To control the spindle by the PMC, the PMC may be required to read the S value specified by the CNC.
If the spindle serial output/spindle analog control function is added (if the PMC can control the spindle), the S-code signals <Fn022 to Fn025> and SF signal <Fn007.2> can be output only when several conditions, determined by the CNC spindle control, are satisfied. In some cases, the signals cannot be used under standard conditions.
Specify the related bits of parameter No. 3705 according to the desired application, then use the S-code and SF signals.

- Twelve code signals corresponding to the S value (output)

Twelve code signals corresponding to S value R01O to R12O <Fn036.0 to Fn037.3> are output to the first spindle motor. The output data is calculated from the results of the CNC spindle control. (See Section SPINDLE SPEED CONTROL.)

Even while a spindle is subject to PMC control, an S command that is issued to the CNC is converted to spindle output data and output.
The SIND signal determines whether the speed output command, issued to the spindle motor, is obtained from the twelve code signals corresponding to the S value, or from the R01I to R12I signals calculated and specified by the PMC.
The use of this signal may simplify PMC ladder processing used to enable PMC spindle control.
Using twelve code signals corresponding to S values R01O to R12O <Fn036.0 to Fn037.3>, the input command status of spindle output control by the PMC is not output.

Example

- Sample application 1)

Controlling the first and second spindles of a lathe system

→ Share the gear stages between the first and second spindles.
(If the first spindle uses two gears, for example, specify parameters Nos. 3743 and 3744, thus enabling the use of gears 3 and 4 for the second spindle.)
Perform the necessary setting to enable control of the first and second spindles by the PMC.
To specify a rotation command for the first spindle, enter the gears for the first spindle in GR1 and GR2 and obtain the data of the twelve code signals corresponding to the S value. Specify the data as the speed output command for the first spindle in the PMC control interface for the first spindle.
To specify a rotation command for the second spindle, enter the gears to be used for the second spindle in GR1 and GR2 and obtain the data of the twelve code signals corresponding to the S value. Specify the data as the speed output command for the second spindle in the PMC control interface for the second spindle.

- Sample application 2)

With the stop position of the serial spindle specified externally, specifying the S value as the angle of the stop position for spindle orientation after the spindle positioning mode has been selected

→ Use the gears that are not being used for the first spindle.
(In this application, gear 4 is used to calculate the spindle position. Set parameters Nos. 3744 to 360.)
Specify the M code used to set the spindle to positioning mode and stop the spindle. Enter gear 4 in GR1 and GR2.
Then, specify a spindle positioning angle with the S command.
(To specify the position of 145 degrees, for example, specify S145;.)
Expression 145/360 4095 is calculated and the result is output to the twelve code signals corresponding to the S value (output signal). Enter the data in external stop position commands SHA00 to SHA11 <Gn078.0 to Gn079.3> and perform the orientation.

Signal

PMC spindle control signals

SIND,SSIN,SGN,<Gn033.7,.6,.5>R01I-R12I<Gn032.0-Gn033.3>: for 1st spindle
SIND2,SSIN2,SGN2,<Gn035.7,.6,.5>R01I2-R12I2<Gn034.0-Gn035.3>: for 2nd spindle
SIND3,SSIN3,SGN3,<Gn037.7,.6,.5>R01I3-R12I3<Gn036.0-Gn037.3>: for 3rd spindle
SIND4,SSIN4,SGN4,<Gn273.7,.6,.5>R01I4-R12I4<Gn272.0-Gn273.3>: for 4th spindle

[Classification] Input signal

[Function] The above signals enable the control of a spindle motor by issuing commands from the PMC.
Both the speed and polarity of the spindle motor (direction of rotation) can be controlled.

The speed command and polarity are usually specified by the CNC. The use of these signals allows the user to select whether the speed and polarity are controlled by the CNC or PMC.
Even if the multi-spindle control is not provided, these signals allow the second and third spindles to be controlled.
When the multi-spindle control and type A are being used (if the bit 2 (MSI) of parameter No. 3709 is set to 0), the signals for the second to fourth spindles cannot be used.

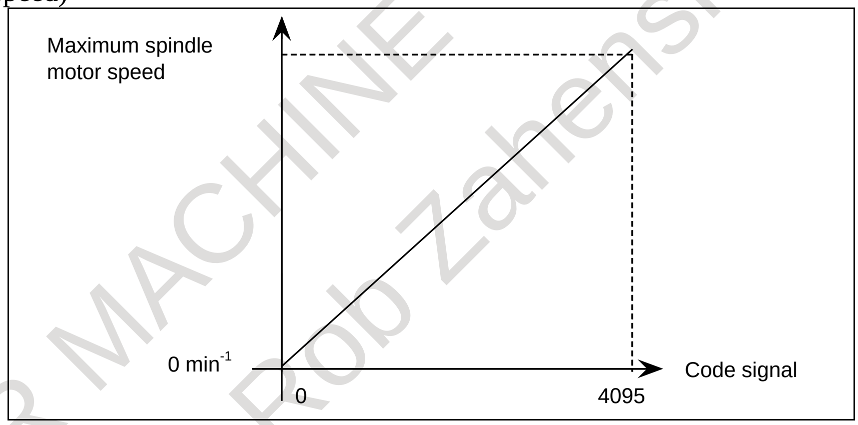
- Details of the signals

- Signal used to select the spindle motor speed command SINDx
  → The above signal is used to select whether the spindle motor speed is controlled by the CNC or PMC.
  "1" : The spindle motor is controlled according to speed commands (R01Ix to R12Ix) issued by the PMC.
  "0" : The spindle motor is controlled according to speed commands issued by the CNC. The spindle speed specified with the S command is output.

- Signals used to input the spindle motor speed command issued by the PMC R01Ix to R12Ix
  → If the PMC is being used to control the spindle motor speed command, specify, in binary format, the value obtained using the following expression.

$$\text{Value to be specified} = \frac{\text{Spindle motor speed}}{\text{Maximum spindle motor speed (parameter No. 4020)}} \times 4095$$

(Spindle motor speed)

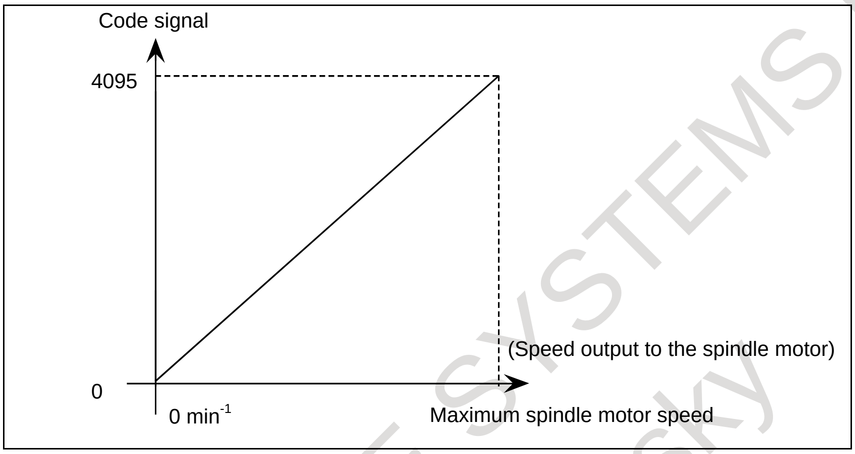


- Signal used to select the polarity of the spindle motor speed command, SSINx
  → The above signal selects whether the output polarity of the spindle motor speed command is controlled by the CNC or PMC.
  "1" : The spindle motor is controlled according to the polarity command (SGNx) issued by the PMC.
  "0" : The CNC controls the polarity. The polarity is determined by the bits 7 (TCW) and 6 (CWM) of parameter No. 3706 and the M03 or M04 command.
- Signal used to specify the polarity of the spindle motor selected by the PMC, SGNx
  → If the PMC is used to control the output polarity of the spindle motor speed command, specify the polarity with this signal.
  "1" : The output polarity of the spindle is negative.
  "0" : The output polarity of the spindle is positive.

Twelve code signals corresponding to the S value
R01O to R12O<Fn036.0 to Fn037.3>

[Classification] Output signal

[Function] The S value, specified in the CNC part program, is converted to the speed output of the spindle motor that is required to control the connected spindle. The converted value is sent to the PMC with twelve code signals, in proportional to the spindle motor speed output.
The speed data, the final result of the CNC spindle control, is output to the spindle motor after the spindle gear ratio, spindle speed override, speed clamp, conversion of the surface speed into the spindle speed by the constant surface speed control command, and other data have been considered.



Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
Gn033	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
Gn034	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2
Gn035	SIND2	SSIN2	SGN2		R12I2	R11I2	R10I2	R09I2
Gn036	R08I3	R07I3	R06I3	R05I3	R04I3	R03I3	R02I3	R01I3
Gn037	SIND3	SSIN3	SGN3		R12I3	R11I3	R10I3	R09I3
Gn272	R08I4	R07I4	R06I4	R05I4	R04I4	R03I4	R02I4	R01I4
Gn273	SIND4	SSIN4	SGN4		R12I4	R11I4	R10I4	R09I4

	#7	#6	#5	#4	#3	#2	#1	#0
Fn036	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
Fn037					R12O	R11O	R10O	R09O

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				ESF
		SFA	NSF					ESF

[Input type] Parameter input

[Data type] Bit path

#0 ESF When the spindle control function (Spindle analog output or Spindle serial output) is used, and the constant surface speed control function is used or bit 4 (GTT) of parameter No. 3706 is set to 1:

0: S codes and SF are output for all S commands.

1: For the T series:

S codes and SF are not output for an S command in the constant surface speed control (G96) mode and a command for maximum spindle speed clamping (G92 S_; (G50 for G code system A)).

For the M series:

S codes and SF are not output for an S command in the constant surface speed control (G96) mode.

> **NOTE**
> The operation of this parameter varies between the T series and M series.
> For the T series:
> This parameter is valid when bit 4 (EVS) of parameter No. 3705 is set to 1.
> For the M series:
> For an S command for maximum spindle speed clamping (G92 S_;), SF is not output, regardless of the setting of this parameter.

#4 EVS When the spindle control function (Spindle analog output or Spindle serial output) is used, S codes and SF are:

0: Not output for an S command.

1: Output for an S command.

The output of S codes and SF for an S command in constant surface speed control mode (G96), or for an S command used to specify maximum spindle speed clamping (G92 S_; (G50 for G code system A)) depends on the setting of bit 0 (ESF) of parameter No. 3705.

#5 NSF For the M series, when a T type gear is selected (with bit 4 (GTT) of parameter No. 3706 set to 1 or enable constant surface speed control (bit 0 (SSC) of parameter No.8133 is 0)), and an S code is specified:

0: SF is output.

1: SF is not output.

> **NOTE**
> This parameter does not affect S code output. For an S command for maximum spindle speed clamping (G92 S_;), SF is not output, regardless of the setting of this parameter.

#6 SFA The SF signal is output:

0: When gears are switched.

1: Irrespective of whether gears are switched.

	#7	#6	#5	#4	#3	#2	#1	#0
3709						MSI		

[Input type] Parameter input
[Data type] Bit path

#2 MSI In multi-spindle control, the signal used to select the spindle motor speed command SIND is valid

0: Only when the first spindle is valid (SIND signal for the 2nd, 3rd spindle becomes ineffective) (TYPE-A)

1: For each spindle irrespective of whether the spindle is selected (Each spindle has its own SIND signal). (TYPE-B)

Note

NOTE

1 If the spindle fails to move after the PMC issues the spindle motor speed command, check the following:
Type A is selected (the bit 2 (MSI) of parameter No. 3709 is set to 0) when the multi-spindle control is used.
→ The second to fourth spindles cannot be controlled. The first spindle can be controlled only when the spindle selection signal SWS1 is set to “1”.
The spindle stop signal for each axis is set to “0” when the multi-spindle control is being used.
→ Spindle stop signal for each axis *SSTPx <G027.3, 4, 5, G026.6> stops the spindle.
M03/M04 is not specified when the CNC is being used to control the output polarity.
→ If the bit 7 (TCW) of parameter No. 3706 is set to 1, the M03/M04 command issued to the CNC changes the output polarity for the spindle motor. If no M03/M04 command is specified after the CNC is turned on, the specified speed output is not sent to the spindle motor because the output polarity has not been determined.

2 The SF signal indicates that output of the S code to the PMC has been completed. The signal does not indicate the end of the command for specifying the spindle speed.

3 For an explanation of connecting the second, third or fourth spindle, see Sections "SPINDLE SERIAL OUTPUT" and "MULTI-SPINDLE CONTROL."

4 If the multi-spindle control is not being used, the CNC does not issue any commands to the second to fourth spindles. The output polarity is controlled by the SGNx signal. It is not affected by the SSINx signal.
The speed output to the spindle motor can be controlled only when the SINDx signal is set to “1”.

10.7 EXTENDED SPINDLE NAME

Overview

Extended spindle names can consist of up to three characters beginning with ‘S’, which is the first spindle name character. Use of extended spindle names allows commands to be issued to a particular spindle without specifying a P command.
For details, see the description of extended spindle names in "MULTI SPINDLE CONTROL".

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	Multi-spindle control

10.8 CONSTANT SURFACE SPEED CONTROL

Overview

When a surface speed (in m/min or feet/min) is specified with an S code (a numeric value following S), the spindle speed is controlled so that the surface speed is kept constant with respect to the change in tool position.

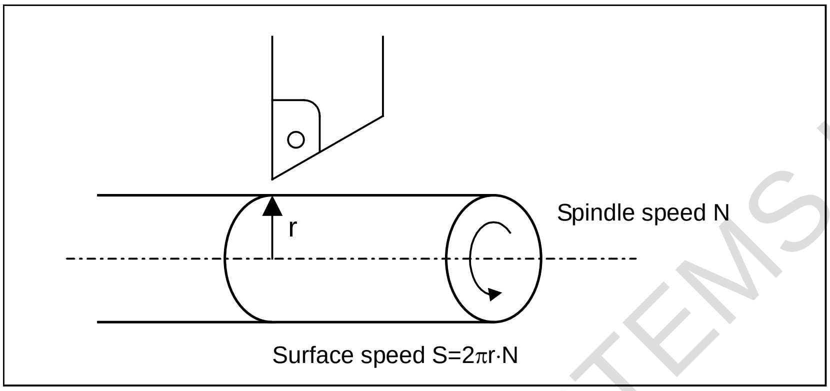


Fig. 10.8 (a) Relationship between surface speed and spindle speed

Explanation

Specify constant surface speed control as follows:

G96 S_ (Surface speed m/min (in metric input) or feet/min (in inch input))

To cancel constant surface speed control, specify the following:

G97 S_ (Spindle speed min^{-1})

To perform constant surface speed control, the maximum spindle speed for each gear select signal issued from the PMC side must be set by parameters Nos. 3741-3744.

The gear select signal is a 2 bit code signal (GR1, GR2). The relationship between the signal and gear number is :

GR1	GR2	Gear number
"0"	"0"	1
"1"	"0"	2
"0"	"1"	3
"1"	"1"	4*

(* :Gear 4 is available only in the lathe specifications.)

- Example of Spindle Analog Output

Assume that gear switching is two stage switching. If the spindle speed with the output 10 V is 1000 min^{-1} for the low speed gear (G1) and 2000 min^{-1} for the high speed gear (G2), set these speeds to the parameters Nos. 3741 and 3742, respectively. In this case, the spindle output has the linear relationship shown below (Fig. 10.8 (b)):

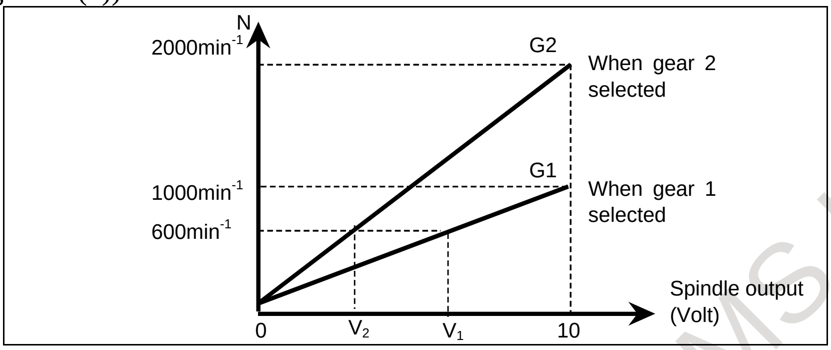


Fig. 10.8 (b)

Here, S = 60 m/min is given as the surface speed; if the position of the present X-axis cutter is 16 mm from the center, the spindle speed N becomes 600 min^{-1} ($S = 2\pi r \cdot N$). Therefore, V_1 (for G1) or V_2 (for G2) is calculated inside the CNC and output to the machine side.

V1 ; 6 (V)
V2 ; 3 (V)

The value of output voltage V is calculated automatically from the following equations:

(i) G96

$V=10S/2\pi rR$

R : Spindle speed (min^{-1}) at 10V output voltage (that is , spindle speed set by parameters Nos. 3741 to 3744)
S : Surface speed (m/min) specified by S command
r : Radius value in the X-axis direction (m)

(ii) G97

$V=10N/R$

R : Spindle speed at 10V output voltage (min^{-1})
N : Spindle speed given by S command (min^{-1})

- Spindle Serial Output

The output to the spindle in spindle serial output is a digital data.
Therefore assume the following relation for calculation:
Spindle analog output (voltage) 10V = Spindle serial output (digital data) 4095.
The above calculation becomes as follows:
The value of Spindle output D:

(i) G96

$D=4095S/2\pi rR$

R : Spindle speed (min^{-1}) at maximum spindle motor speed (that is , spindle speed set by parameters Nos. 3741 to No. 3744)
S : Surface speed (m/min) specified by S
r : Radius value in the X-axis direction (m)

(ii) G97

$D=4095N/R$

R : Spindle speed at maximum spindle motor speed (min^{-1})
N : Spindle speed given by S command (min^{-1})

- Constant surface speed control with no position coder

With a machine having (or using) no position coder, constant surface speed control cannot usually be performed during feed per revolution, but is enabled by setting bit 1 (FPR) of parameter No. 3729 or bit 2 (PCL) of parameter No. 1405 to 1.

NOTE
1 This function is not effective to spindle output control by the PMC.
2 When the setting of bit 1 (FPR) of parameter No.3729 or bit 2 (PCL) of parameter No.1405 is 1, feed per revolution cannot be used to spindle output control by the PMC.

For example, the following program is executed with G code system B and diameters specified.

```
N1   G00 X50.Z10. ;
N2   G96 G95 S12 ;        ←Constant surface speed control and feed per revolution begin.
N3   G01 X20.Z30. F10.;
N4   Z50. ;
N5   G97 S200;            ←Constant surface speed control ends.
N6   G00 Z200.;
N7   M30 ;
```

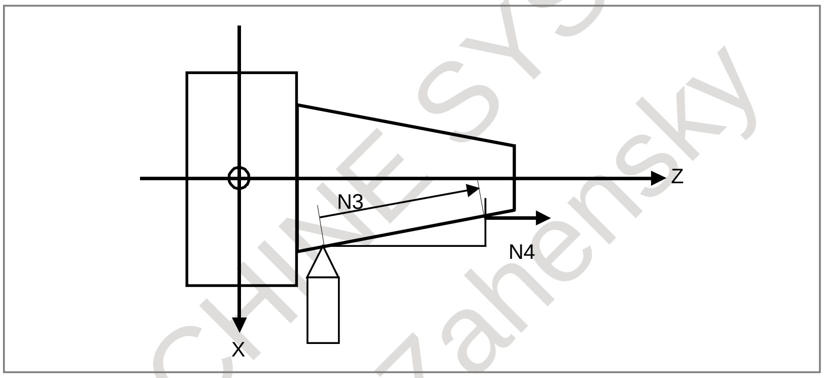


Fig. 10.8 (c) Feed per revolution without position coder

In this program, block N2 issues a constant surface speed control command (G96), a surface speed command (S12 m/min), and a feed-per-revolution command (G95). Block N3 causes the CNC to change the spindle speed specification from 76.4 min^{-1} to 191 min^{-1} so as to maintain a constant surface speed during movement to X = 20.
Meanwhile, the feed-per-revolution speed is changed according to the changed spindle speed specification, and used for movement along the feed axis. However, the specified spindle speed is clamped to the upper limit to the spindle speed specification (for the first spindle, using parameter No. 3772). In the above program, the feed-per-revolution speed command F10 in block N3 corresponds to an actual speed of 764 (mm/min) to 1910 (mm/min).

- Command for maximum spindle speed clamping

When rotation at the maximum spindle speed set by the parameter causes a problem in constant surface speed control, execute a command (M series: G92S_; T series: G50S_;) for maximum spindle speed clamping before starting constant surface speed control in order to prevent the spindle from rotating at a speed equal to or greater than a certain rotation speed.
If bit 4 (CSA) of parameter No. 3712 is set to 1, it is possible to prevent the spindle from rotating at an unintended speed by issuing alarm PS5557, “NO MAX SP SPEED CLAMP COMMAND”, if constant surface speed control is specified when a command for maximum spindle speed clamping has not been executed after power-on due to a program error or an operation error such as execution of a machining program in an incorrect order.

- Spindle speed calculation during the 3-dimensional coordinate system conversion / the tilted working plane indexing

In the constant surface speed control in the 3-dimensional coordinate system conversion / the tilted working plane indexing mode, the coordinate system which is the standard of speed calculation can be changed by the parameter 3CS(No.11221#6).

Table 10.8 (a) The standard coordinate system of spindle speed calculation

Parameter 3CS	Coordinate system of standard
0	Workpiece coordinate
1	Program coordinate (3-dimensional coordinate system conversion) / Feature coordinate (Tilted working plane indexing)

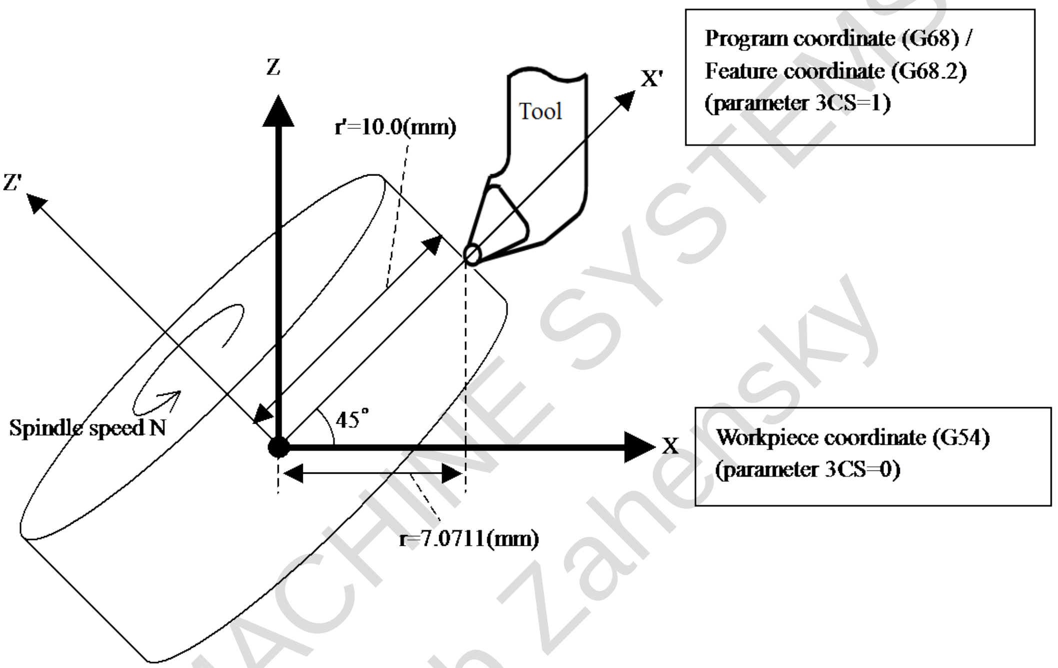


Fig.10.8 (d) Spindle speed calculation during the 3-dimensional coordinate system conversion / the tilted working plane indexing

Example)
When the followings are set by the programming.
- the surface speed S=10(m/min)
- the axis as the calculation reference is X

```
N1 G68 X0 Y0 Z0 I0 J1 K0 R-45.0
N2 G01 G90 X0 Y0 Z0 F100.0
N3 G96 S10 P1
N4 X10.0 Y0 Z0
N5 G97
N6 G69
```

Table 10.8 (b) The spindle speed in N4 (calculation by relation of N=S/2πr)

Parameter 3CS	Radius value r (mm) In the axis (X) as the calculation reference	Spindle speed N(mm^{-1})
0	7.0711	225
1	10.0	159

- Multi-spindle control

Clamp of maximum spindle speed

When bit 0 (CLM) of parameter No.3785 is set to 0, and spindle selection by extended spindle name is enabled, if extended spindle name is commanded at clamp command of maximum spindle speed G92 (G50 in G code system A of T series), alarm PS0539, "MAX SP SPEED CLAMP COMMAND ERROR" is issued. "S" has to be commanded at clamp command of maximum spindle speed.

Example)
Suppose that extended spindle name is set as follows.

Table 10.8 (c)

Spindle number	Extended spindle name
First spindle	S
Second spindle	SB

If "G92 SB30 ;" is commanded, alarm PS0539, "MAX SP SPEED CLAMP COMMAND ERROR" is issued. Clamp of maximum spindle speed has to be commanded "G92 S30 ;". If "S" is set as extended spindle name, alarm PS0539 is not issued.

Selection of axis as the calculation reference in constant surface speed control

When bit 0 (CLM) of parameter No.3785 is set to 0, and spindle selection by address P or extended spindle name of multi-spindle control is enabled, selection of axis as the calculation reference in constant surface speed control "G96 P_ ;" is commanded, alarm PS0190, "ILLEGAL AXIS SELECTED (G96)" is issued. Please change the setting of parameter No.3770 using programmable parameter input (G10) or the like in changing axis as the calculation reference in constant surface speed control.

Example
Suppose that spindle selection by address P is enabled.
- If "G96 P_ ;" is commanded, alarm PS0190 is issued.
- If "G96 S_ P_ ;" is commanded, spindle selected by address P is controlled by constant surface speed control.

Example
Suppose that extended spindle name is set as Table 11.7 (c).
- If "G96 P_ ;" is commanded, alarm PS0190 is issued.
- If "G96 S_ P_ ;" , or "G96 SB_ P_ " is commanded, alarm PS0190 is issued.

When extended spindle name is enabled, address P cannot be commanded at G96 block.

G96 command without surface speed

When bit 1 (G96) of parameter No.3785 is set to 1, and spindle selection by address P or extended spindle name is enabled, if G96 command without surface speed is commanded, alarm PS5355, "S CODE IS NOT COMMANDED AT G96" is issued.

> **NOTE**
> When spindle selection by address P or extended spindle name is enabled, if G96 is commanded without surface speed, surface speed which is commanded at previous G96 command is applied. Axis as the calculation reference in constant surface speed control which is commanded previously regardless of G96/G97 mode is applied. In order to prevent rotation of unexpected spindle by G96 command without surface speed, please command surface speed at G96 command.

- System variable

Clamp value of spindle maximum speed

System variable number	System variable name	Attribute	Description
#100959	[#_CSSSMAX]	R	Spindle maximum speed commanded at clamp command of maximum spindle speed

CNC reads block ahead of the executing block. The value of this system variable can be read after being executed clamp command of maximum spindle speed.

Example 1)

```
O1234 ;
N10 G92 S5000 ;
N20 G96 S60 ;
N30 G01 X200.0 Z-300.0 F2.0 ;
    :
    :
N60 G92 S2500 ;
N70 #1 = #100959 ;
```

Suppose that N30 block is executing. And suppose that CNC has read up to N70 block. Because N60 block has not been executed, maximum spindle speed which can be read at N70 block is 5000[min^{-1}] commanded at N10 block. Therefore, 5000 is substituted for #1. In program of example 1, if 2500 has to be substituted for #1, command block preventing buffering (such as M code preventing buffering) before N70 block.

In the following cases, #100959 is -1.

- Clamp command of maximum spindle speed G92 (G50 in G code system A of T series) has not been executed after turning on the power.

- Rigid tapping

If rigid tapping is commanded during constant surface speed control, alarm PS0200, "ILLEGAL S CODE COMMAND" is issued. Command rigid tapping after canceling constant surface speed control.

Signal

Gear selection signal (Input) GR2, GR1<Gn028.2,.1>

Refer to Section, "SPINDLE SPEED CONTROL."

Constant surface speed signal CSS<Fn002.2>

[Classification] Output signal
[Function] This signal indicates that constant surface speed control is in progress.
[Output cond.] "1" indicates that constant surface speed control mode (G96) is in progress, while "0" indicates it is not.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn002						CSS		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1405						PCL		

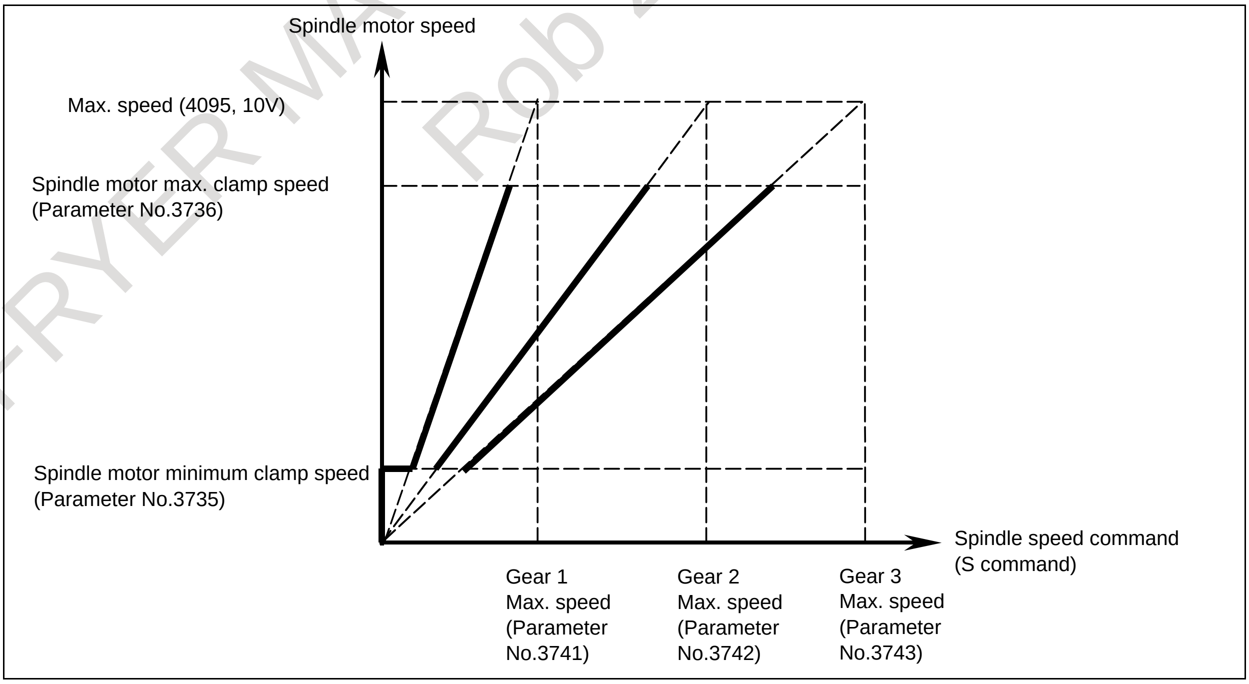
[Input type] Parameter input
[Data type] Bit path

#2 PCL The function for constant surface speed control without the position coder is:
0: Not used.
1: Used.

> **NOTE**
> 1 Enable constant surface speed control (bit 0 (SSC) of parameter No.8133 is 0).
> 2 This function is not effective to spindle output control by the PMC.
> 3 When this parameter is set to 1, feed per revolution cannot be used to spindle output control by the PMC.

3741	Maximum spindle speed for gear 1
3742	Maximum spindle speed for gear 2
3743	Maximum spindle speed for gear 3
3744	Maximum spindle speed for gear 4

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999
Set the maximum spindle speed corresponding to each gear.



	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				ESF
			NSF					ESF

[Input type] Parameter input
[Data type] Bit path

#0 ESF When the spindle control function (Spindle analog output or Spindle serial output) is used, and the constant surface speed control function is used or bit 4 (GTT) of parameter No. 3706 is set to 1:

0: S codes and SF are output for all S commands.

1: For the T series:

S codes and SF are not output for an S command in the constant surface speed control (G96) mode and a command for maximum spindle speed clamping (G92 S_; (G50 for G code system A)).

For the M series:

S codes and SF are not output for an S command in the constant surface speed control (G96) mode.

> **NOTE**
> The operation of this parameter varies between the T series and M series.
> For the T series:
> This parameter is valid when bit 4 (EVS) of parameter No. 3705 is set to 1.
> For the M series:
> For an S command for maximum spindle speed clamping (G92 S_;), SF is not output, regardless of the setting of this parameter.

#4 EVS When the spindle control function (Spindle analog output or Spindle serial output) is used, S codes and SF are:

0: Not output for an S command.

1: Output for an S command.

The output of S codes and SF for an S command in constant surface speed control mode (G96), or for an S command used to specify maximum spindle speed clamping (G92 S_; (G50 for G code system A)) depends on the setting of bit 0 (ESF) of parameter No. 3705.

#5 NSF For the M series, when a T type gear is selected (with bit 4 (GTT) of parameter No. 3706 set to 1 or enable constant surface speed control (bit 0 (SSC) of parameter No.8133 is 0)), and an S code is specified:

0: SF is output.

1: SF is not output.

> **NOTE**
> This parameter does not affect S code output. For an S command for maximum spindle speed clamping (G92 S_;), SF is not output, regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3706								
				GTT				

[Input type] Parameter input
[Data type] Bit path

#4 GTT Spindle gear selection method is:
0: Type M.
1: Type T.

NOTE
1 M type
The gear selection signal is not input. The CNC selects a gear based on the speed range of each gear set by a parameter beforehand according to S codes, and the selected gear is posted by outputting the gear selection signal. Moreover, the spindle speed matching the gear selected by the output gear selection signal is output.
T type
The gear selection signal is input. The spindle speed matching the gear selected by this signal is output.
2 When enable constant surface speed control (bit 0 (SSC) of parameter No.8133 is 0), type T is selected, regardless of whether this parameter is specified.
3 When type T spindle gear switching is selected, the following parameters have no effect:
No.3705#2(SGB), No.3751, No.3752,
No.3705#1(GST), No.3705#3(SGT), No.3761, No.3762,
No.3705#6(SFA), No.3735, No.3736
On the other hand, parameter No. 3744 becomes usable.

	#7	#6	#5	#4	#3	#2	#1	#0
3708			SOC					

[Input type] Parameter input
[Data type] Bit path

#5 SOC During constant surface speed control (G96 mode), the speed clamp by the maximum spindle speed clamp command (G92 S_; (G50 for G code system A of lathe system)) is carried out:
0: Before spindle speed override.
1: After spindle speed override.
If this parameter is set to 0, the spindle speed may exceed the maximum spindle speed (numeric value following S in G92 S_; (G50 for G code system A of lathe system)).
If this parameter is set to 1, the spindle speed is limited to the maximum spindle speed.
The spindle speed is limited to the upper limit of spindle speed specified in parameter No. 3772, irrespective of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3709							RSC	

[Input type] Parameter input
[Data type] Bit path

#1 RSC In the constant surface speed control mode, the surface speed of a rapid traverse block is calculated:
0: In accordance with the coordinates of the end point.
1: In accordance with the current value, as in cutting feed.

	#7	#6	#5	#4	#3	#2	#1	#0
3712				CSA				

[Input type] Parameter input
[Data type] Bit

#4 CSA When the constant surface speed control command (G96S_) is issued, if no command for maximum spindle speed clamping (G92S_; in system M or G50S_; in system T) has been executed since power turn-on:
0 : No alarm is issued (conventional specification).
1 : Alarm PS5557, “NO MAX SP SPEED CLAMP COMMAND”, is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
3729							FPRs	

[Input type] Parameter input
[Data type] Bit spindle

#1 FPRs Feed per revolution (without a position coder) is:
0: Not used for a spindle.
1: Used for a spindle.
In a machine that does not use a position coder, when bit 1 (FPRs) of parameter No.3729 is set to 1 for each axis, feed per revolution can be performed with a spindle command. A feed per revolution is specified with G95 (G99 for lathe systems) in the same way as for normal operation.
When multi-spindle control is performed, the target spindle for feed per revolution is selected with a position coder select signal (PC2SLC<Gn028.7>, PC3SLC<Gn026.0>, PC4SLC <Gn026.1>).

NOTE
1 The option for constant surface speed control is required.
2 This function is not effective to spindle output control by the PMC.
3 When this parameter is set to 1, feed per revolution cannot be used to spindle output control by the PMC.

3770	Axis as the calculation reference in constant surface speed control

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the axis as the calculation reference in constant surface speed control.

NOTE
When 0 is set to parameter No.3770, constant surface speed control is always applied to the X-axis. In this case, specifying P in a G96 block has no effect on the constant surface speed control.

3771	Minimum spindle speed in constant surface speed control mode (G96)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

Set the minimum spindle speed in the constant surface speed control mode (G96).
When constant surface speed control is performed and the spindle speed is lower than the speed given by this parameter, the spindle speed is clamped to the speed given by this parameter.

3772	Maximum spindle speed

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

This parameter sets the maximum spindle speed.
When a command specifying a speed exceeding the maximum speed of the spindle is specified, or the speed of the spindle exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

⚠ **CAUTION**

1 When 0 is set in this parameter, the speed of the spindle is not clamped.
2 When spindle speed command control is applied using the PMC, this parameter has no effect, and the spindle speed is not clamped.

NOTE

1 For M series, this parameter is valid if enable constant surface speed control (bit 0 (SSC) of parameter No.8133 is 0).
2 When enable constant surface speed control (bit 0 (SSC) of parameter No.8133 is 0), the spindle speed is clamped at the maximum speed, regardless of whether the G96 mode or G97 mode is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
3785							G96	CLM

[Input type] Parameter input
[Data type] Bit

#0 CLM When spindle selection by address P or extended spindle name of multi-spindle control is enabled, constant surface speed control command is:

0: The following specifications.
- If extended spindle name is commanded in clamp command of maximum spindle speed, alarm PS0539, “MAX SP SPEED CLAMP COMMAND ERROR” is issued.
- If address P is commanded at selection of axis as the calculation reference in constant surface speed control, alarm PS0190, “ILLEGAL AXIS SELECTED (G96)” is issued.

1: Conventional specifications.

#1 G96 When spindle selection by address P in multi-spindle control, or extended spindle name is enabled, if G96 is commanded without surface speed :
0: Alarm PS5355, " S CODE IS NOT COMMANDED AT G96" is not issued.
1: Alarm PS5355 is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
11221		3CS						

[Input type] Parameter input
[Data type] Bit path

#6 3CS The spindle speed calculation of constant surface speed control during the 3-dimensional coordinate system conversion / the tilted working plane indexing is :
0: Based on the workpiece coordinate system.
1: Based on the program coordinate system (3-dimensional coordinate system conversion) / the feature coordinate system (Tilted working plane indexing).

Alarm and message

Number	Message	Description
PS0190	ILLEGAL AXIS SELECTED (G96)	An illegal value was specified in P in a G96 block or parameter No. 3770. When spindle selection by address P or extended spindle name of multi-spindle control is enabled, selection of axis as the calculation reference in constant surface speed control "G96 P_ ;" cannot be commanded.
PS0539	MAX SP SPEED CLAMP COMMAND ERROR	Clamp of maximum spindle speed is illegal. Extended spindle name cannot be commanded for clamp of maximum spindle speed.
PS5355	S CODE IS NOT COMMANDED AT G96	S code is not commanded at G96. Command S code at G96 block.
PS5557	NO MAX SP SPEED CLAMP COMMAND	No command for maximum spindle speed clamping (M series: G92S_; T series: G50S_;) was executed before the constant surface speed control command (G96S_) was started. Execute a command for maximum spindle speed clamping.

Caution

⚠ **CAUTION**

1 If the spindle speed corresponding to the calculated surface speed exceeds the speed specified in the spindle speed clamp command (G50S_ for T series and G92S_ for M series) during the G96 mode, the actual spindle speed is clamped at the value specified in the spindle speed clamp command.
If the specified spindle speed is lower than the value specified in parameter No. 3771, the actual spindle speed is clamped at the speed specified by No. 3771.

2 If the constant surface speed control function is provided for a machining center system, it affects gear change under normal spindle control. (See "SPINDLE SPEED CONTROL.")

Note

> **NOTE**
> Simultaneous use of multi-spindle control enables constant surface speed control for spindles other than the first spindle. (See "MULTI-SPINDLE CONTROL.")

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Constant surface speed control (G96, G97)

10.9 ACTUAL SPINDLE SPEED OUTPUT (T series)

Overview

The PMC can read actual spindle speed.

Signal

Actual spindle speed signals

AR00 to AR15<Fn040, Fn041> : First spindle
AR002 to AR152<Fn202, Fn203> : Second spindle
AR003 to AR153<Fn206, Fn207> : Third spindle
AR004 to AR154<Fn272, Fn273> : Fourth spindle

[Classification] Output signal

[Function] These 16-bit binary code signals output from the CNC to the PMC the actual spindle speed obtained by feedback pulses from the position coder mounted on the spindle.

[Output cond.] These 16-bit binary code signals are always output. The actual spindle speed and the signals have the following relationship:

$$Spindle\ speed = \sum_{i=0}^{15} \left\{ 2^i \times V_i \right\} \mathrm{min}^{-1}$$

where Vi = 0 when ARi is 0 and Vi = 1 when ARi is 1

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn040	AR07	AR06	AR05	AR04	AR03	AR02	AR01	AR00
Fn041	AR15	AR14	AR13	AR12	AR11	AR10	AR09	AR08
Fn202	AR072	AR062	AR052	AR042	AR032	AR022	AR012	AR002
Fn203	AR152	AR142	AR132	AR122	AR112	AR102	AR092	AR082
Fn206	AR073	AR063	AR053	AR043	AR033	AR023	AR013	AR003
Fn207	AR153	AR143	AR133	AR123	AR113	AR103	AR093	AR083
Fn272	AR074	AR064	AR054	AR044	AR034	AR024	AR014	AR004
Fn273	AR154	AR144	AR134	AR124	AR114	AR104	AR094	AR084

Note

> **NOTE**
> An absolute error of about 0.5 min^{-1} exists as a measuring error.
> If the speed exceeds 65535min^{-1}, 65535 is assumed and output.

10.10 SPINDLE POSITIONING

Overview

This function positions the spindle using the spindle motor and position coder.
The function has a coarser least command increment compared with the Cs contour control function and has no interpolation capability with other axes. However, it can be installed more easily because the position detector is a position coder.
Generally, the spindle positioning axes are clamped mechanically except when positioning is under way.

Explanation

In the turning process, the workpiece is rotated by the spindle to which it is attached, at the speed specified for the spindle motor. This spindle control state is called the spindle rotation mode.
The workpiece mounted on the spindle can be positioned at a certain angle by moving the spindle attached to the spindle motor by the certain angle. This function is called the spindle positioning function. This spindle control state is called the spindle positioning mode.

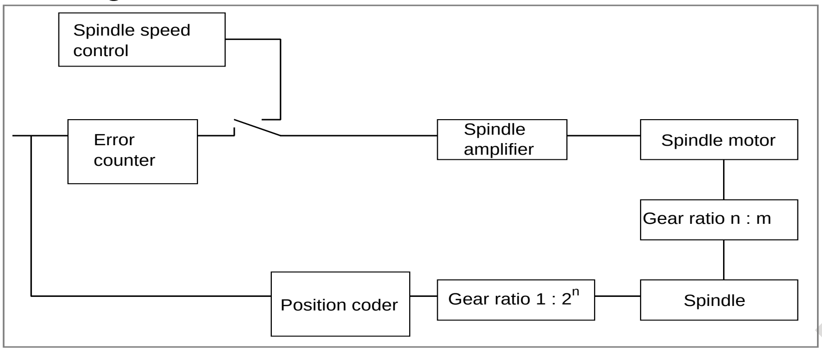
The spindle positioning function can perform the following operations:

- Release the spindle rotation mode and enter the spindle positioning mode
  Specifying a particular M code (set in the relevant parameter) sets a reference position in the spindle positioning mode. (spindle orientation)
- Position the spindle in the spindle positioning mode
  1) Position an optional angle using axis address,
  2) Position a semi-fixed angle using a specific M code (set in the relevant parameter).
- Release the spindle positioning mode and enter the spindle rotation mode
  Specifying a particular M code (set in the relevant parameter) changes the spindle to the spindle rotation mode.

The least command increment, least input increment, and maximum value for the spindle positioning axis are as follows:

- Least command increment
  360/4096 ≒ 0.088 deg (When the gear ratio of the spindle to position coder is 1:1)
- Least input increment
  0.001 deg (IS-B)
- Max. programmable dimension
  ±999999.999 deg

- Control block diagram



The spindle positioning function is enabled only when the number of pulses of the position coder is 4096 pulses (If parameter No.3716#7=1 and serial spindle is used, a number of pluses other than 4096 pluses is possible) and the gear ratio of the spindle to position coder is one of the ratios listed below.

$1 : 2^n$ （n : Integer not smaller than 0）

The least command increment (detection unit) for each gear ratio is as follows (Table 10.10 (a)):

Table 10.10 (a)

Gear ratio of spindle to position coder	Least command increment (detection unit) deg
1 : 1	0.088 (1×360/4096)
1 : 2	0.176 (2×360/4096)
1 : 4	0.352 (4×360/4096)
1 : 8	0.703 (8×360/4096)
⋮	⋮
1 : N	(N×360/4096)

- Selecting a spindle positioning axis

As the target axis for spindle positioning, any axis address can be set in parameter No. 1020.
When setting the servo axis number of a spindle positioning axis (in parameter No. 1023), add a minus sign (-) to the spindle number for which spindle positioning is to be performed.
Spindle positioning axes are assigned after the end of controlled axes.

Setting example 1)
Servo axis : X, Y, Z
Spindle positioning axis : C (first spindle)
Number of spindles : 1

Number of controlled axes	Parameter No. 1020	Parameter No. 1023
1	88 (X)	1
2	89 (Y)	2
3	90 (Z)	3
4	67 (C)	-1

Setting example 2)
Servo axis : X, Z
Spindle positioning axis : C (first spindle), B (second spindle)
Number of spindles : 2

Number of controlled axes	Parameter No. 1020	Parameter No. 1023
1	88 (X)	1
2	90 (Z)	2
3	67 (C)	-1
4	66 (B)	-2

> **NOTE**
> If bit 4 (KSV) of parameter No.11802 is set to 1, spindle positioning will be disabled.

- Switching to spindle positioning mode

Switching to the spindle positioning mode is specified using the M code set in parameter No. 4960.
When the spindle motor is used for the ordinary spindle operation (the spindle rotation mode), switching to the spindle positioning mode must be made before spindle positioning can be performed.

Spindle orientation

After switching to the spindle positioning mode is completed, a spindle orientation is performed. The orientation function stops the spindle at a predetermined position.

The orientation direction is specified by using bit 5 (ZMI) of parameter No. 1006 for analog spindles or bit 4 (RETSV) of parameter No. 4000 for serial spindles.
With the grid shift function, the orientation position can be shifted within ±180 degrees by setting parameter No. 1850 when the spindle is an analog spindle, or can be shifted within the range from 0 to 360 degrees by setting parameter No. 4073 when the spindle is a serial spindle.

Orientation speed

When an analog spindle is used, a movement is made at the rapid traverse rate used for reference position return, which is set in parameter No. 1428, until the orientation enable speed is attained, and after the one-rotation signal is detected, orientation is performed at the FL feedrate set in parameter No. 1425.

* Orientation enable speed

$$1024 \geq \left(\frac{F \times P}{60 \times G \times 0.088} \right) \geq 128\,(pulses)$$

Set the orientation enable speed so that the expression shown is satisfied:
F : Rapid traverse rate for reference position return (deg/min)
G : Position gain (1/sec)
P : Gear ratio of the spindle to position coder

If the above condition is not satisfied, the one-rotation signal cannot be detected, so the spindle continues moving at the rapid traverse rate.
The rapid traverse rate for reference position return cannot be overridden.

* FL speed
Set the FL feedrate so that the following expression is satisfied:

$$1024 \geq \left(\frac{FL \times P}{60 \times G \times 0.088} \right) \geq effective\,area\,of\,spindle\,(pulses)$$

FL : Feedrate (deg/min)
G : Position gain (1/sec)
P : Gear ratio of the spindle to position coder

If the above condition is not satisfied, the accurate stop position cannot be detected, so the orientation is started again from the rapid traverse operation.

When a serial spindle is used, the orientation speed is determined by parameter settings on the spindle side.
When an orientation is performed for a serial spindle, the spindle motor rotates several turns and then the spindle is stopped at the orientation position.

Omitting orientation

When the orientation at the time of switching to the spindle positioning mode is not required (for example, when no start position is specified, and only incremental positioning from the current position is needed), the orientation performed at the time of switching to the spindle positioning mode can be omitted by setting bit 2 (ISZ) of parameter No. 4950. In this case, when the M code for switching to the spindle positioning mode is specified, the spindle control mode is just changed to the spindle positioning mode without orientation. This completes the switching to the spindle positioning mode.

Program origin

The position at which the orientation is completed is regarded as the program origin. The program origin can be changed by coordinate system setting (G92 (T series G code B, C/M series) or G50 (T series G code A)) or automatic coordinate system setting (bit 0 (ZPR) of parameter No. 1201).
When the orientation is omitted, the program origin is not established, so operations by absolute commands in spindle positioning by axis address are not guaranteed.

- Command system

The command system comes in two types: The first positions a semi-fixed angle; the second positions an optional angle.

Semi-fixed angle positioning

Specify a positioning angle with an M code. Six types of M codes Mα to M(α+5) can be specified. The value of α is set in parameter No. 4962 in advance. The positioning angles corresponding to Mα to M(α+5) are listed below, and β is set in parameter No. 4963 in advance.

M code (Example) β=α+5	Indexing angle	(Example) β=30°
Mα	β	30°
M (α+1)	2β	60°
M (α+2)	3β	90°
M (α+3)	4β	120°
M (α+4)	5β	150°
M (α+5)	6β	180°

When the number (γ) of M codes used is specified in parameter No. 4964, M codes Mα to M(α+(γ-1)) can be specified in up to 255 ways (Mα to M(α+(255-1))).

M code (Example) γ = 11	Indexing angle	(Example) β=30°
Mα	β	30°
M (α+1)	2β	60°
M (α+2)	3β	90°
M (α+3)	4β	120°
to	to	to
M (α+11-1)	11β	330°

One of the rotation directions can be set by setting bit 1 (IDM) of parameter No. 4950.

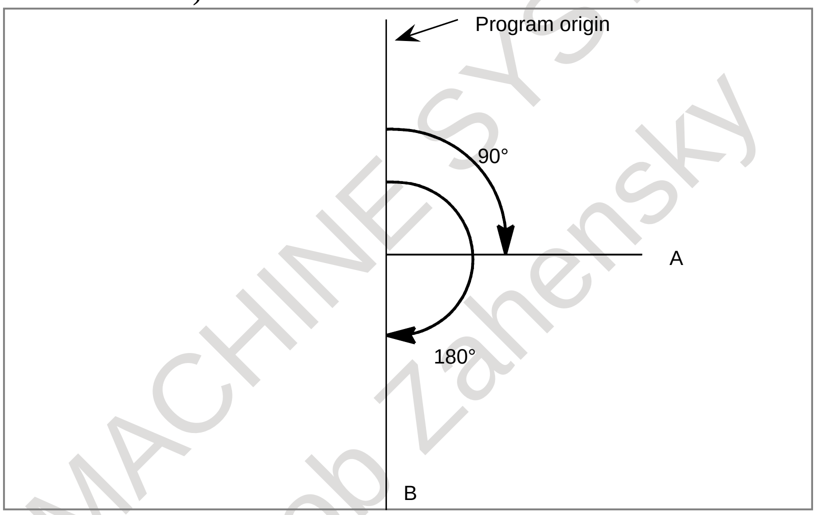
Optional angle positioning

The position at any angle is specified by using an axis address followed by a signed numeric value. The axis address must be specified in the G00 mode.
(In the subsequent explanation, the axis address is assumed to be the C-axis.)
(Example)
C-45000
C180.000
A numeric value with a decimal point can be entered. The position of the decimal point denotes the degree position.
(Example) C36.0 = C36

Absolute and incremental commands

Positioning by specifying a semi-fixed angle (by M code) is always incremental.
One of the rotation directions can be set by setting bit 1 (IDM) of parameter No. 4950.

For positioning at any angle, an absolute command and an incremental command can be specified.
When an absolute command is used for positioning at any angle, short-cut control (by setting bit 1 (RAB) of parameter No. 1008 to 0) is also enabled using the rotation axis rollover function (by setting bit 0 (ROA) of parameter No. 1008 to 1).



Command method		G code system A for T series		G code system B, C for T series M series	
		Address used	Command of A-B on the above Figure	Address used and G-code	Command of A-B on the above Figure
Absolute command	Direct the end point position by the distance from the program origin	C	C180.0 ;	G90, C	G90C180.0 ;
Incremental command	Command by the distance between the start and end points.	H	H90.0 ;	G91, C	G91C90.0 ;

Spindle positioning feedrate

Spindle positioning is done at the rapid traverse rate specified by parameter No. 1420.
Overrides of 100%, 50%, 25% and F0 (parameter No. 1421) are also applied.

- Spindle positioning reset

A specific M code (parameter No. 4961) must be specified when the mode is changed from spindle positioning to normal spindle rotation.

Also in the following cases, the spindle positioning mode is canceled, and the spindle rotation mode is entered:

<1> A reset operation (including emergency stop) is performed when a servo alarm is issued.
<2> A reset operation (including emergency stop) is performed when a spindle alarm is issued.
<3> The orientation operation is stopped in the middle by a reset, alarm, and so on.
<4> A reset operation (including emergency stop) is performed when bit 0 (IOR) of parameter No. 4950 is set to 1.

Diagnosis data

1544	Spindle positioning sequence status

The spindle positioning sequence status is indicated. (For each spindle)

1) Sequence for switching to the spindle positioning mode

Indication	Description
00010003	Wait until spindle stop complete signal SPSTPs is turned on (“0”->”1”)
00010005	Wait for completion of switching to spindle positioning mode
00010006	Wait until spindle unclamp completion signals *SUCPFs is turned on (“1”->”0”)
00010008	Wait for completion of orientation operation
00010009	Wait for completion of preset processing after orientation
0001000A	Wait until spindle clamp completion signals *SCPFs is turned on (“1”->”0”)

2) Spindle positioning sequence

Indication	Description
00020004	Wait until SPSTPs is turned on (“0”->”1”)
	Wait until *SUCPFs is turned on (“1”->”0”)
00020006	Wait for completion of positioning operation
00020007	Wait until *SCPFs is turned on (“1”->”0”)

For spindle positioning sequence B specifications (when bit 7 (IMB) of parameter No. 4950 is set to 1)

00040003	Wait until SPSTPs is turned on (“0”->”1”)
00040005	Wait for completion of switching to spindle positioning mode
00040006	Wait until *SUCPFs is turned on (“1”->”0”)
00040008	Wait for completion of positioning operation
00040009	Wait until *SCPFs is turned on (“1”->”0”)
0004000A	Wait for completion of spindle positioning mode cancellation

3) Spindle positioning mode cancellation sequence

Indication	Description
00030003	Wait until SPSTPs is turned on (“0”->”1”)
	Wait until *SUCPFs is turned on (“1”->”0”)
00030004	Wait for completion of spindle positioning mode cancellation

5207	Clamp/unclamp sequence status

The clamp/unclamp sequence status is indicated. (For each controlled axis)

1) Sequence for switching to the spindle positioning mode

Indication	Description
00030001	Wait until spindle stop complete signal SPSTPs is turned on (“0”->”1”)
00030002	Wait for completion of switching to spindle positioning mode
00030004	Wait until spindle unclamp completion signals *SUCPFs is turned on (“1”->”0”)
00030005	Wait for completion of orientation operation
00030006	Wait until spindle clamp completion signals *SCPFs is turned on (“1”->”0”)

2) Spindle positioning sequence

Indication	Description
00070001	Wait until SPSTPs is turned on (“0”->”1”)
00070002	Wait until *SUCPFs is turned on (“1”->”0”)
00070003	Wait for completion of positioning operation
00070004	Wait until *SCPFs is turned on (“1”->”0”)

For spindle positioning sequence B specifications (when bit 7 (IMB) of parameter No. 4950 is set to 1)

000D0001	Wait until SPSTPs is turned on (“0”->”1”)
000D0002	Wait for completion of switching to spindle positioning mode
000D0004	Wait until *SUCPFs is turned on (“1”->”0”)
000D0005	Wait for completion of positioning operation
000D0006	Wait until *SCPFs is turned on (“1”->”0”)
000D0007	Wait for completion of spindle positioning mode cancellation

3) Spindle positioning mode cancellation sequence

Indication	Description
000B0001	Wait until SPSTPs is turned on (“0”->”1”)
000B0002	Wait until *SUCPFs is turned on (“1”->”0”)
000B0004	Wait for completion of spindle positioning mode cancellation
FFFFFFFF	End of sequence

Spindle tuning screen

In the spindle positioning mode, indications related to the spindle positioning mode appear.
For details, please refer to Spindle tuning screen of MAINTENANCE MANUAL (B-64695EN).

Signal

Spindle unclamp signals

SUCLPA<Fn038.1> : First spindle
SUCLPB<Fn400.1> : Second spindle
SUCLPC<Fn400.2> : Third spindle
SUCLPD<Fn400.3> : Fourth spindle

[Classification] Output signal

[Function] These signals specify that spindle mechanical clamping be released in a spindle positioning sequence.
When this signal turns to “1”, unclamp the spindle on the machine (release the brakes or extract the pin).

[Output cond.] Refer to the sequence (Timing chart) in this section.

Spindle unclamp completion signals
***SUCPFA<Gn028.4> : First spindle**
***SUCPFB<Gn400.1> : Second spindle**
***SUCPFC<Gn400.2> : Third spindle**
***SUCPFD<Gn400.3> : Fourth spindle**

[Classification] Input signal

[Function] These signals indicate that unclamping the spindle is complete in response to the spindle unclamp signal SUCLPs.

Spindle clamp signals
SCLPA<Fn038.0> : First spindle
SCLPB<Fn401.1> : Second spindle
SCLPC<Fn401.2> : Third spindle
SCLPD<Fn401.3> : Fourth spindle

[Classification] Output signal

[Function] These signals specify that the spindle be clamped mechanically in a spindle positioning sequence.
When this signal turns to "1", clamp the spindle on the machine (apply the brakes or insert the pin).

[Output cond.] Refer to the sequence (Timing chart) in this section.

Spindle clamp completion signals
***SCPFA<Gn028.5> : First spindle**
***SCPFB<Gn401.1> : Second spindle**
***SCPFC<Gn401.2> : Third spindle**
***SCPFD<Gn401.3> : Fourth spindle**

[Classification] Input signal

[Function] These signals indicate that clamping the spindle is complete in response to the spindle clamp signal SCLPs.

Spindle positioning mode signals
MSPOSA<Fn039.0> : First spindle
MSPOSB<Fn402.1> : Second spindle
MSPOSC<Fn402.2> : Third spindle
MSPOSD<Fn402.3> : Fourth spindle

[Classification] Output signal

[Function] These signals indicate that the spindle is in the spindle positioning mode.

[Output cond.] Refer to the sequence (Timing chart) in this section.

Spindle stop complete signal
SPSTPA<Gn028.6> : First spindle
SPSTPB<Gn402.1> : Second spindle
SPSTPC<Gn402.2> : Third spindle
SPSTPD<Gn402.3> : Fourth spindle

[Classification] Input signal

[Function] After checking that these signals are set to 1, the CNC performs spindle orientation, spindle positioning, and spindle positioning cancellation.

- Signals related to gear change

Gear selection signals GR1O,GR2O,GR3O<Fn034.0 to 2>

[Classification] Output signal
[Function] These signals inform the PMC of the selected gear stage.
[Output cond.] For information about these signals, see "SPINDLE SPEED CONTROL".

Gear selection signals GR1, GR2<Gn028.1,.2>

[Classification] Input signal
[Function] These signals inform the CNC of the gear stage currently selected.
[Operation] For information about these signals, see "SPINDLE SPEED CONTROL".

Clutch/gear signals (serial spindle)

CTH1A,CTH2A<Gn070.3,Gn070.2> : First spindle
CTH1B,CTH2B<Gn074.3,Gn074.2> : Second spindle
CTH1C,CTH2C<Gn204.3,Gn204.2> : Third spindle
CTH1D,CTH2D<Gn266.3,Gn266.2> : Fourth spindle

See the relevant manual of the serial spindle.

These signals determine the gear parameters (such as the loop gain) to be used.
Although the gear selection signals for a serial spindle are CTH1 and CTH2, also input GR1 and GR2. Avoid changing the states of these signals in the spindle positioning mode.

Relationship between selected gear and spindle gear selection signals

CNC side							Serial spindle side	
Lathe system and machining center system (with constant surface speed control (SSC(No.8133#0)=1))			Machining center system (with constant surface speed control (SSC(No.8133#0)=0))					
GR2	GR1	Selected gear	GR3O	GR2O	GR1O	Selected gear	CTH1	CTH2
0	0	1st gear	0	0	0	1st gear	0	0
0	1	2nd gear	0	1	1	2nd gear	0	1
1	0	3rd gear	1	0	0	3rd gear	1	0
1	1	4th gear					1	1

NOTE

1 With a machining system, GR1 and GR2 need not be input if the constant surface speed control is not available (SSC(No.8133#0)=0), and bit 4 (GTT) of parameter No. 3706 is 0. When the CNC outputs gear selection signals GR3O, GR2O, and GR1O to make a gear change on the machine side, CTH1 and CHT2 must also be input.

2 The above combinations of clutch/gear signals (CTH1 and CTH2) are just examples.
Since the serial spindle selects a gear by using CTH1 and CTH2 independent of gear selection on the CNC, the signals must be input as necessary, and serial spindle parameters must be set accordingly.

- Other signals

Reference position return completion signal ZPx<Fn094>

[Classification] Output signal
[Function] This signal indicates that spindle orientation of the spindle positioning axis is completed.
[Output cond.] This signal is set to 1 when spindle orientation is completed, and the signal is set to 0 when spindle positioning or spindle positioning cancellation is performed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn028		SPSTPA	*SCPFA	*SUCPFA				
Gn028						GR2	GR1	
Gn400					*SUCPFD	*SUCPFC	*SUCPFB	
Gn401					*SCPFD	*SCPFC	*SCPFB	
Gn402					SPSTPD	SPSTPC	SPSTPB	
Fn034						GR3O	GR2O	GR1O
Fn038							SUCLPA	SCLPA
Fn039								MSPOSA
Fn094	ZP8	ZP7	ZP6	ZP5	ZP4	ZP3	ZP2	ZP1
Fn400					SUCLPD	SUCLPC	SUCLPB	
Fn401					SCLPD	SCLPC	SCLPB	
Fn402					MSPOSD	MSPOSC	MSPOSB	

- For first serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn070					CTH1A	CTH2A		

- For second serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn074					CTH1B	CTH2B		

- For third serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn204					CTH1C	CTH2C		

- For fourth serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn266					CTH1D	CTH2D		

- Sequence (Timimg chart)

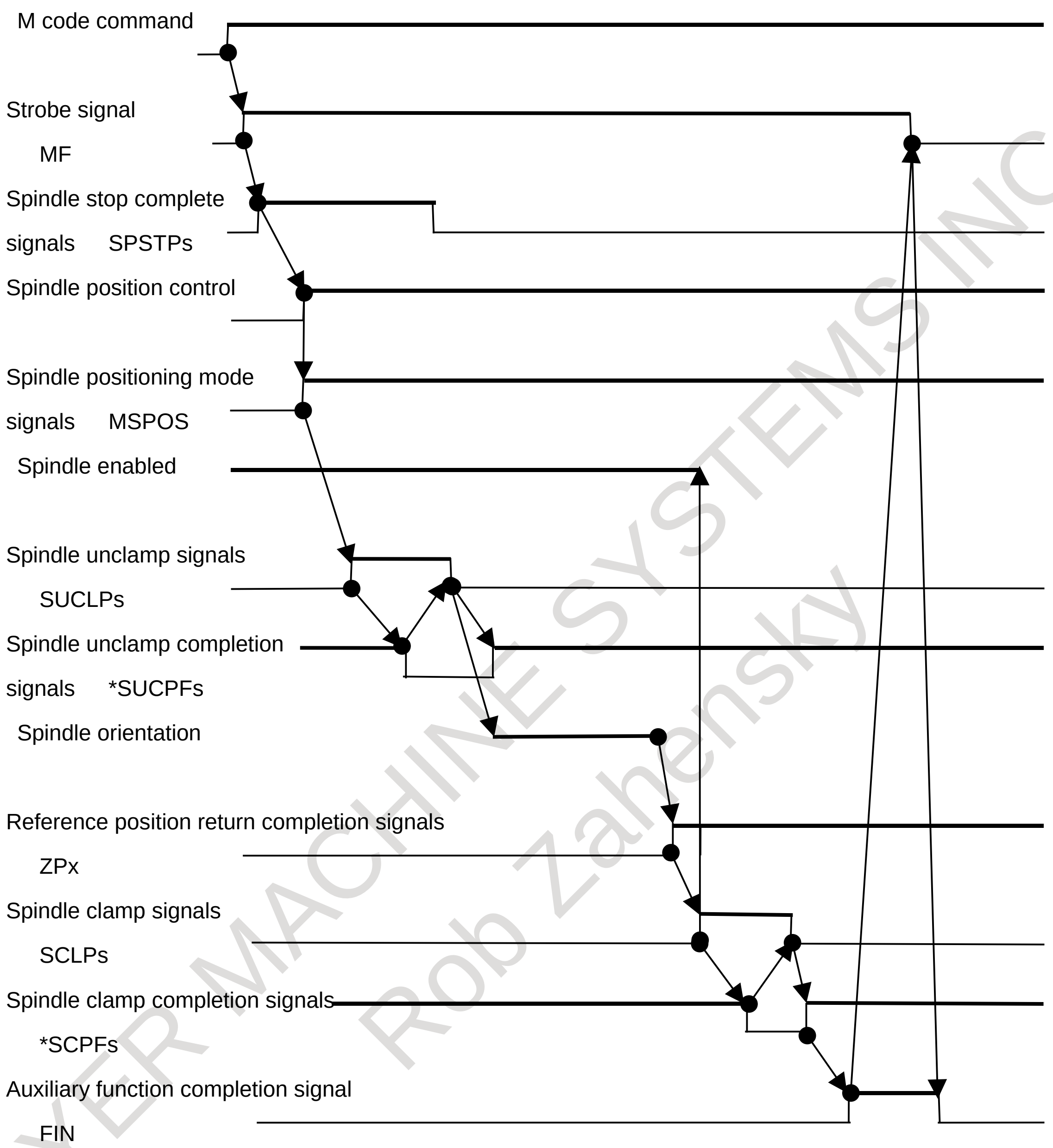
Switching to the spindle positioning mode

The operation of canceling the spindle rotation mode to enter the spindle positioning mode is performed by specifying an M code for switching to the spindle positioning mode.
Set the M code for switching to the spindle positioning mode in parameter No. 4960 in advance.

<1> Suppose that an M code is specified by a program.
<2> Auxiliary function code signal M00 to M31 is sent, and the auxiliary function strobe signal MF is set to 1 at the same time. This sending procedure is the same as for ordinary auxiliary functions.
<3> When strobe signal MF is set to 1, the PMC must read the code signal and perform corresponding processing. After completing the corresponding processing, the PMC sets spindle stop complete signal SPSTPs to 1 when the spindle is stopped.

<4> As soon as setting strobe signal MF to 1, the CNC checks whether spindle stop complete signal SPSTPs is set to 1. When SPSTPs is set to 1, the CNC changes the spindle control mode from the spindle rotation mode to spindle positioning mode, and sets spindle positioning mode signal MSPOSs to 1.
<5> The CNC sets spindle unclamp signal SUCLPs to 1.
<6> When spindle unclamp signal SUCLPs is set to 1, the PMC performs processing to make the spindle usable for positioning. When completing the processing, the PMC sets spindle unclamp completion signal *SUCPFs to 0.
<7> When spindle unclamp completion signal *SUCPFs are set to 0, the CNC sets spindle unclamp signal SUCLPs to 0 to post that the CNC has received the *SUCPFs signal.
<8> When spindle unclamp signal SUCLPs is set to 0, the PMC must set spindle unclamp completion signal *SUCPFs to 1.
<9> The CNC rotates the spindle and stops the spindle at a certain position. (This is generally called spindle orientation.) Then, the CNC sets reference position return completion signal ZPx (x denotes the controlled axis number of the spindle positioning axis) to 1.
<10>The CNC sets spindle clamp signal SCLPs to 1.
<11>When spindle clamp signal SCLPs is set to 1, the PMC clamps the spindle mechanically with a device such as a clutch or shot pin as necessary. After completing the clamping operation, the PMC sets spindle clamp completion signal *SCPFs to 0.
<12>When spindle clamp completion signal *SCPFs is set to 0, the CNC sets spindle clamp signal SCLPs to 0 to post that the CNC has received *SCPFs.
<13>When spindle clamp signal SCLPs is set to 0, the PMC sets spindle clamp completion signal *SCPFs to 1.
<14>Finally, the PMC sets completion signal FIN to 1 for auxiliary function strobe signal MF. In response to this, the CNC sets the MF signal to 0. Then, the PMC sets the FIN signal to 0.

Timing chart for switching to spindle positioning mode



⚠ **CAUTION**

1 Switching between the spindle rotation mode and spindle positioning mode must be performed on the machine side in response to M code commands issued from the CNC.
When the spindle positioning mode is canceled, the spindle becomes ready for accepting speed commands such as an S command. Therefore, a PMC ladder program must be created so that spindle rotation is not performed when the spindle is clamped on the machine side.
Example:
When the spindle is clamped on the machine side, spindle stop signal *SSTP <G029.6> is set to 0.

> ⚠ **CAUTION**
>
> 2 When the spindle is clamped on the machine side (by using a device such as a clutch or shot pin), the spindle must be disabled. Enabling or disabling the spindle means to separately instruct the spindle control unit to enable or disable spindle motor operation by the PMC ladder and so on.
> Example: When a serial spindle is enabled, SFRA <G070.5> is set to 1. When the serial spindle is disabled, SFRA <G070.5> is set to 0.
> Refer to the manual of the spindle control unit to be connected, and add instructions to the ladder program.

Spindle positioning

In the spindle positioning mode, positioning is performed using one of the following two programmed commands:

1) Positioning at a semi-fixed angle by specifying an M code
2) Positioning at any angle by specifying an axis address

The operations performed by using the above two specification methods are the same, except that in positioning at any angle, input/output signals related to auxiliary functions are not exchanged.
The M code used for positioning at a semi-fixed angle is set in parameters Nos. 4962 and 4964 in advance.

When positioning at a semi-fixed angle is performed by specifying the M code, specification A or B can be selected by setting bit 7 (IMB) of parameter No. 4950.

If specification B is selected, specifying an M code that specifies the spindle positioning angle performs operations (1) to (3) below successively.

(1) Canceling the spindle rotation mode and switching to the spindle positioning mode
(2) Positioning the spindle in the spindle positioning mode
(3) Canceling the spindle positioning mode and switching to the spindle rotation mode

If specification A is selected, only the spindle positioning operation is performed in the spindle positioning mode.

[Specification A: Bit 7 (IMB) of parameter No. 4950 = 0]

<1> Suppose that an M code is specified by a program. If an M code is specified in the spindle rotation mode, the CNC issues alarm PS0224, "ZERO RETURN NOT FINISHED".

<2> Auxiliary function code signal M00 to M31 is sent, and the auxiliary function strobe signal MF is set to 1 at the same time. This sending procedure is the same as for ordinary auxiliary functions.

<3> When strobe signal MF is set to 1, the PMC must read the code signal and perform corresponding processing. After completing the corresponding processing, the PMC sets spindle stop complete signal SPSTPs to 1 when the spindle is stopped.

<4> As soon as setting strobe signal MF to 1, the CNC checks whether spindle stop complete signal SPSTPs is set to 1. When SPSTPs is set to 1, the CNC sets the spindle unclamp signal SUCLPs to 1.

<5> When spindle unclamp signal SUCLPs is set to 1, the PMC unclamps the spindle, and when unclamping of the spindle is completed, the PMC sets spindle unclamp completion signal *SUCPFs to 0.

<6> When spindle unclamp completion signal *SUCPFs is set to 0, the CNC sets spindle unclamp signal SUCLPs to 0 to post that the CNC has received the *SUCPFs signal.

<7> When spindle unclamp signal SUCLPs is set to 0, the PMC must set spindle unclamp completion signal *SUCPFs to 1. After setting the SUCLPs signal to 0, the CNC rotates the spindle, makes a movement as specified, then stops the spindle.

<8> The CNC sets spindle clamp signal SCLPs to 1.

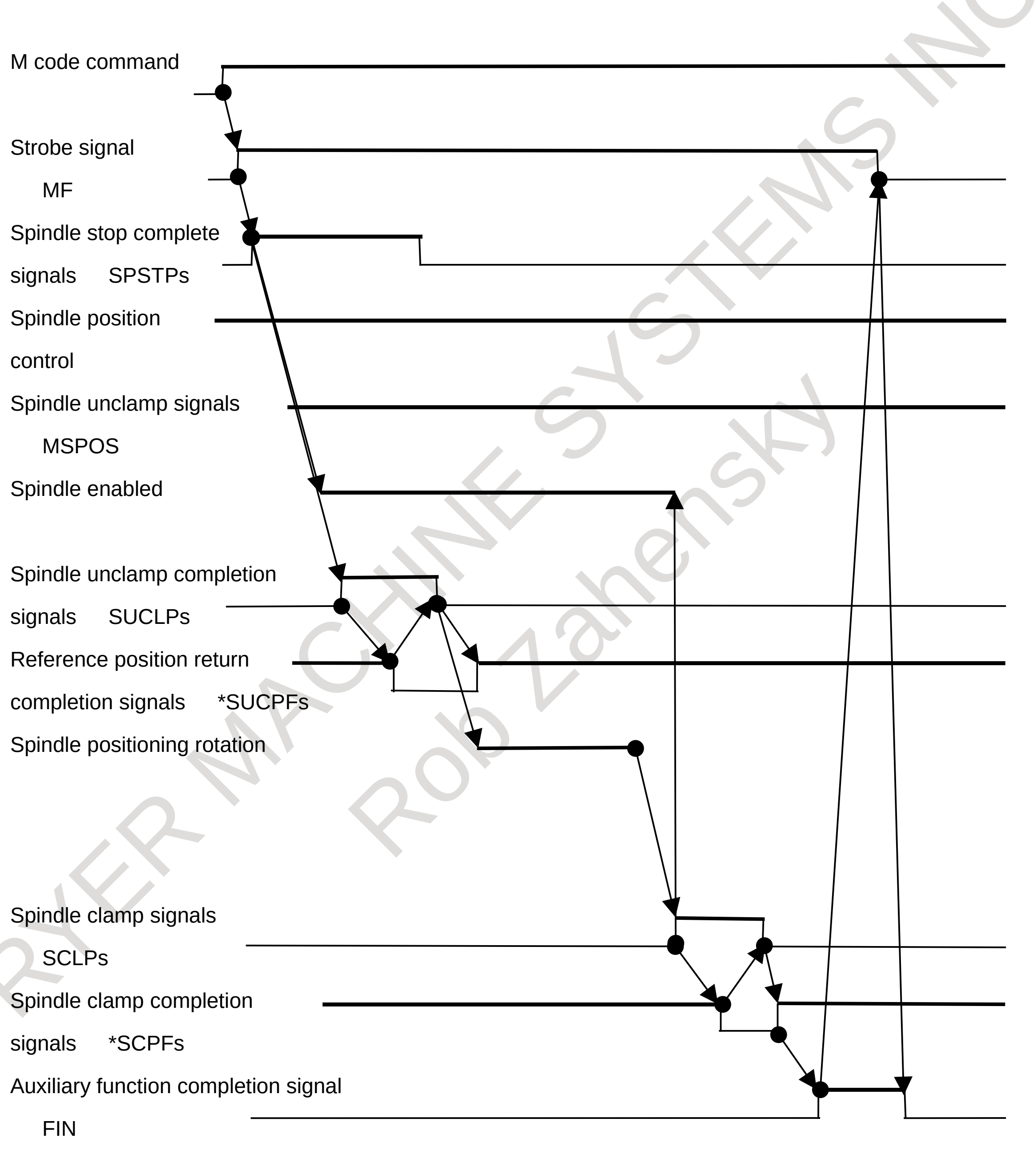
<9> When spindle clamp signal SCLPs is set to 1, the PMC clamps the spindle mechanically with a device such as a clutch or shot pin as necessary. After completing the clamping operation, the PMC sets spindle clamp completion signal *SCLPFs to 0.

<10>When spindle clamp completion signal*SCLPs is set to 0, the CNC sets spindle clamp signal SCLPs to 0 to post that the CNC has received *SCLPFs

<11>When spindle clamp signal SCLPs is set to 0, the PMC sets spindle clamp completion signal *SCLPFs to 1.

<12>Finally, the PMC sets completion signal FIN to 1 for auxiliary function strobe signal MF. In response to this, the CNC sets the MF signal to 0. Then, the PMC sets the FIN signal to 0.

Timing chart for spindle positioning (specification A)



[Specification B: Bit 7 (IMB) of parameter No. 4950 = 1]

<1> Suppose that an M code is specified in a program.

<2> Auxiliary function code signal M00 to M31 is sent, and the auxiliary function strobe signal MF is set to 1 at the same time. This sending procedure is the same as for ordinary auxiliary functions.

<3> When strobe signal MF is set to 1, the PMC must read the code signal and perform corresponding processing. After completing the corresponding processing, the PMC sets spindle stop complete signal SPSTPs to 1 when the spindle is stopped.

<4> As soon as setting strobe signal MF to 1, the CNC checks whether spindle stop complete signal SPSTPs is set to 1. When the SPSTPs signal is set to 1, the CNC switches the spindle control mode from the spindle rotation mode to spindle positioning mode, and sets spindle positioning mode signal MSPOSs to 1.

<5> The CNC sets spindle unclamp signal SUCLPs to 1.

<6> When spindle unclamp signal SUCLPs is set to 1, the PMC unclamps the spindle, and when unclamping of the spindle is completed, the PMC sets spindle unclamp completion signal *SUCPFs to 0.

<7> When spindle unclamp completion signal *SUCPFs is set to 0, the CNC sets spindle unclamp signal SUCLPs to 0 to post that the CNC has received the *SUCPFs signal.

<8> When spindle unclamp signal SUCLPs is set to 0, the PMC sets spindle unclamp completion signal *SUCPFs to 1.

<9> After setting the SUCLPs signal to 0, the CNC rotates the spindle, makes a movement as specified, then stops the spindle.

<10>The CNC sets spindle clamp signal SCLPs to 1.

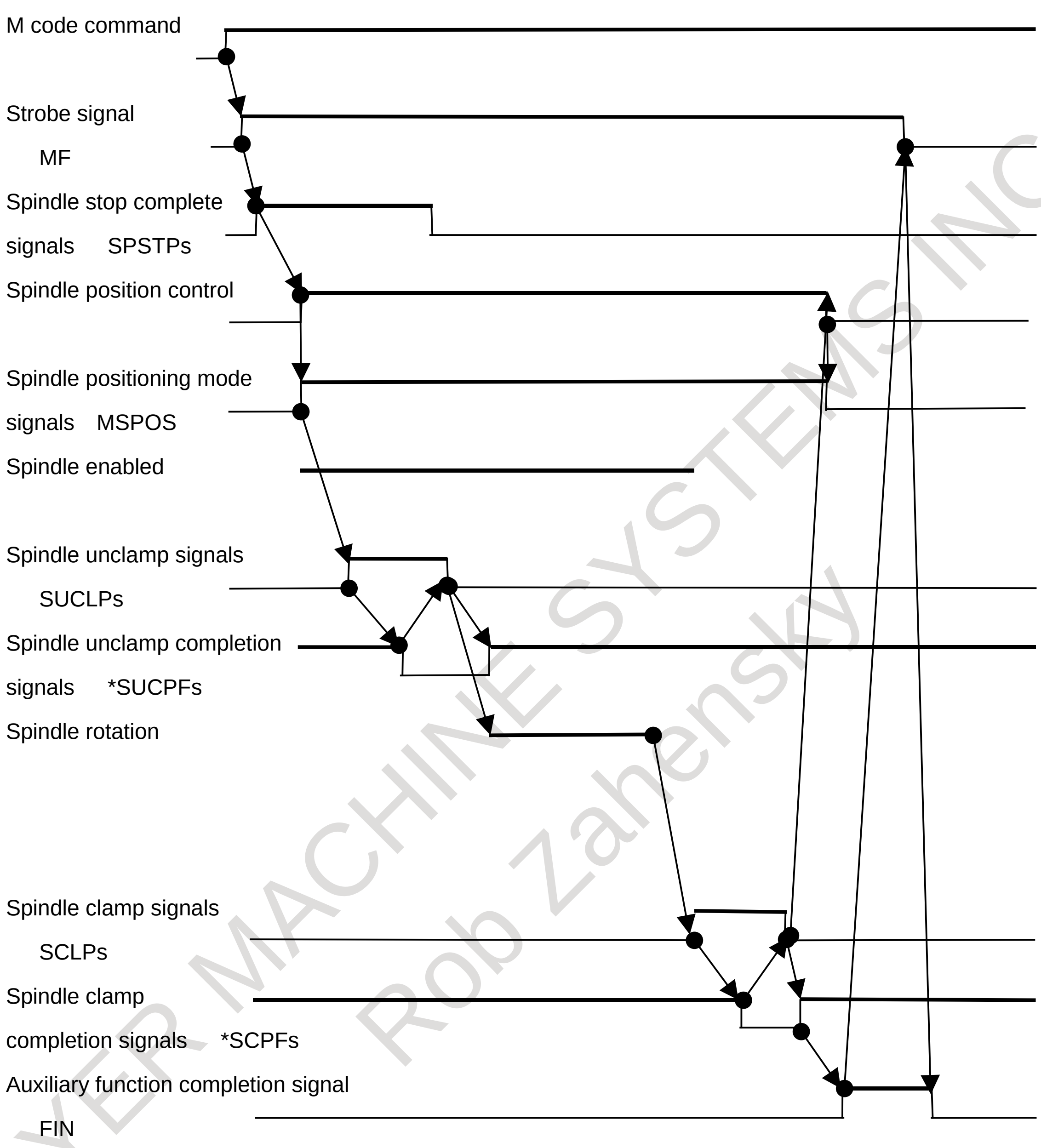
<11>When spindle clamp signal SCLPs is set to 1, the PMC clamps the spindle mechanically with a device such as a clutch or shot pin as necessary. After completing the clamping operation, the PMC sets spindle clamp completion signal *SCLPFs to 0.

<12>When spindle clamp completion signal *SCLPs is set to 0, the CNC sets spindle clamp signal SCLPs to 0 to post that the CNC has received *SCLPFs. The CNC sets the SCLPs signal to 0, and switches the spindle control mode from the spindle positioning mode to spindle rotation mode at the same time. The CNC sets spindle positioning mode signal MSPOSs to 0.

<13>When spindle clamp signal SCLPs is set to 0, the PMC sets spindle clamp completion signal *SCLPFs to 1.

<14>Finally, the PMC sets completion signal FIN to 1 for auxiliary function strobe signal MF. In response to this, the CNC sets the MF signal to 0. Then, the PMC sets the FIN signal to 0.

Timing chart for spindle positioning (specification B)



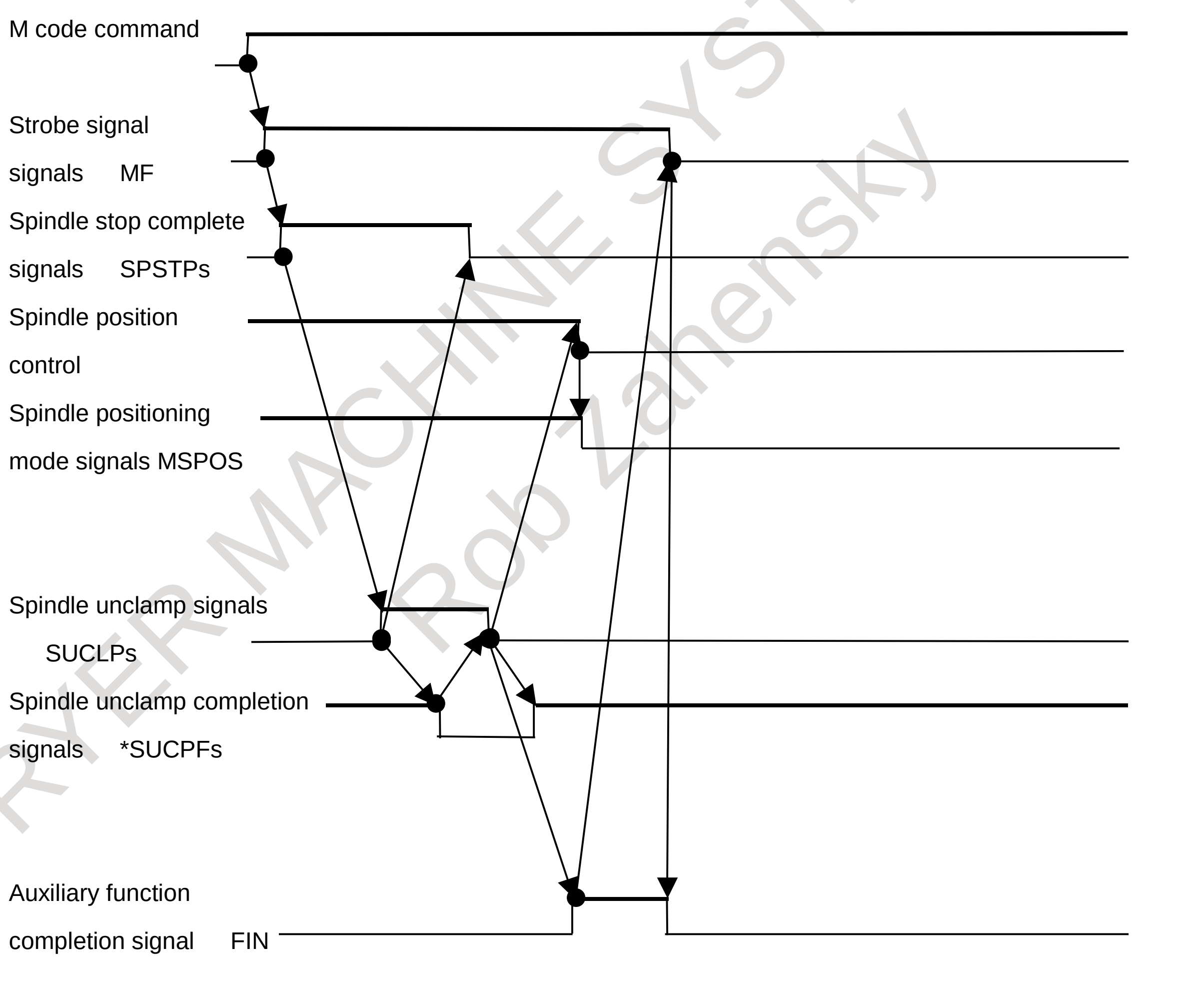
Canceling spindle positioning

To cancel the spindle positioning mode and enter the spindle rotation mode again, use an M code for canceling the spindle positioning mode.
Set the M code in parameter No. 4961 in advance.

<1> Suppose that an M code is specified in a program.
<2> Auxiliary function code signal M00 to M31 is sent, and the auxiliary function strobe signal MF is set to 1 at the same time. This sending procedure is the same as for ordinary auxiliary functions.
<3> When strobe signal MF is set to 1, the PMC must read the code signal and perform corresponding processing. After completing the corresponding processing, the PMC sets spindle stop complete signal SPSTPs to 1 when the spindle is stopped.

<4> As soon as setting strobe signal MF to 1, the CNC checks whether spindle stop complete signal SPSTPs is set to 1. When the SPSTPs signal is set to 1, the CNC sets spindle unclamp signal SUCLPs to 1.
<5> When spindle unclamp signal SUCLPs is set to 1, the PMC unclamps the spindle, and when unclamping of the spindle is completed, the PMC sets spindle unclamp completion signal *SUCPFs to 0.
<6> When spindle unclamp completion signal *SUCPFs is set to 0, the CNC sets spindle unclamp signal SUCLPs to 0 to post that the CNC has received the *SUCPFs signal. As soon as setting the SUCLPs signal to 0, the CNC switches the spindle control mode from the spindle positioning mode to spindle rotation mode. The CNC sets spindle positioning mode signal MSPOSs to 0.
<7> When spindle unclamp signal SUCLPs is set to 0, the PMC sets spindle unclamp completion signal *SUCPFs to 1.
<8> Finally, the PMC sets completion signal FIN to 1 for auxiliary function strobe signal MF. In response to this, the CNC sets the MF signal to 0. Then, the PMC sets the FIN signal to 0.

Timing chart for canceling spindle positioning mode



Parameter

Major related parameters are described below.

	#7	#6	#5	#4	#3	#2	#1	#0
1005								ZRNx

[Input type] Parameter input
[Data type] Bit axis

#0 ZRNx If a move command other than G28 is specified by automatic operation when no reference position return is performed yet after the power is turned on:
0: The alarm PS0224, "ZERO RETURN NOT FINISHED" is issued.
1: Operation is performed without issuing an alarm.

> **NOTE**
> 1 The state in which a reference position has not been established refers to the following state:
> - When an absolute position detector is not used and reference position return has not been performed even once after power-up
> - When an absolute position detector is used and the association of the machine position with the position detected with the absolute position detector has not been completed (See the description of bit 4 (APZx) of parameter No. 1815.)
>
> 2 When the Cs axis coordinates are to be set up, set ZRN to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMIx		DIAx		ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx Setting linear or rotation axis.
#1 ROSx

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No. 1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.

ROSx	ROTx	Meaning
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

#3 DIAx The move command for each axis is based on:
0: Radius specification
1: Diameter specification

#5 ZMIx The direction of manual reference position return is:
0: + direction
1: - direction

	#7	#6	#5	#4	#3	#2	#1	#0
1013							ISCx	ISAx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ISAx
#1 ISCx Increment system of each axis

Increment system	Bit 1 (ISC)	Bit 0 (ISA)
IS-A	0	1
IS-B	0	0
IS-C	1	0

<u>IS-B is specified for the spindle positioning axis.Therefore, set ISA to ISE to 0.</u>

1022	Setting of each axis in the basic coordinate system

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

To determine a plane for circular interpolation, cutter compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane), specify which of the basic three axes (X, Y, and Z) is used for each control axis, or a parallel axis of which basic axis is used for each control axis.
A basic axis (X, Y, or Z) can be specified only for one control axis.
Two or more control axes can be set as parallel axes for the same basic axis.

Setting	Meaning
0	Rotary axis (Neither the basic three axes nor a parallel axis)
1	X axis of the basic three axes

Setting	Meaning
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

In general, the increment system and diameter/radius specification of an axis set as a parallel axis are to be set in the same way as for the basic three axes.

1023	Number of the servo axis for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] -4 to 80

This parameter associates each control axis with a specific servo axis. Specify values 1+8n, 2+8n, 3+8n, 4+8n, 5+8n, and 6+8n (n = 0, 1, 2, …, 9) like 1, 2, 3, 4, 5, …, 77, and 78.
Set the spindle number targeted for spindle positioning preceded by a minus sign (-1 to –4), as the servo axis number of an axis used as the spindle positioning axis.
Settings
-1 : Spindle positioning by using the first spindle
-2 : Spindle positioning by using the second spindle
-3 : Spindle positioning by using the third spindle
-4 : Spindle positioning by using the fourth spindle

NOTE
One spindle cannot be set as multiple spindle positioning axes.

1260	The shift amount per one rotation of a rotary axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the shift amount per one rotation of a rotary axis.
For the rotary axis used for cylindrical interpolation, set the standard value.
For the spindle positioning axis, set 360.0.

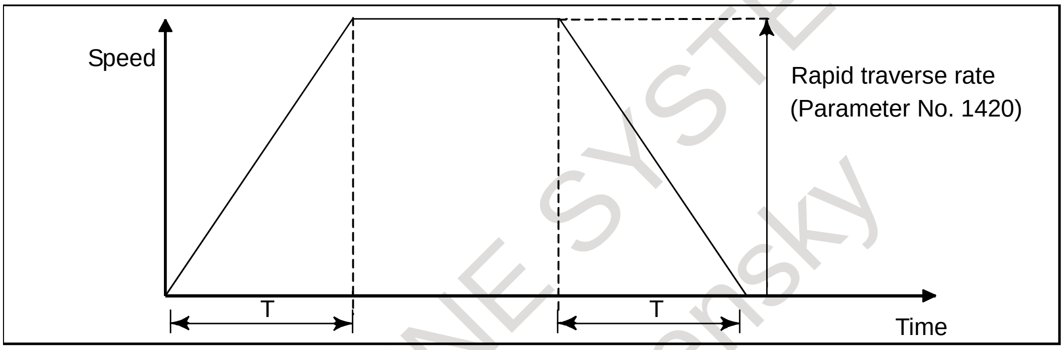
1420	Rapid traverse rate for each axis

[Input type] Parameter input

[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

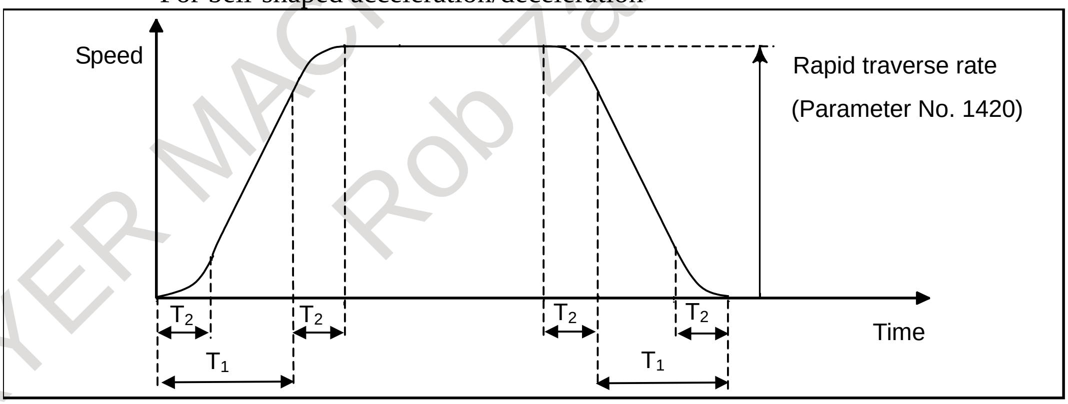
1620	**Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
Specify a time constant used for acceleration/deceleration in rapid traverse.
[Example]
For linear acceleration/deceleration



T : Setting of parameter No. 1620

For bell-shaped acceleration/deceleration



T_1 : Setting of parameter No. 1620
T_2 : Setting of parameter No. 1621
(However, $T_1 \geq T_2$ must be satisfied.)
Total acceleration (deceleration) time : $T_1 + T_2$
Time for linear portion : $T_1 - T_2$
Time for curve portion : $T_2 \times 2$

1820	Command multiplier for each axis (CMR)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 1 to 96
Set 2 for spindle positioning axis.

1821	Reference counter size for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999
Set a reference counter size.
As a reference counter size, specify a grid interval for reference position return based on the grid method.
When a value less than 0 is set, the specification of 10000 is assumed.

1826	In-position width for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
The in-position width is set for each axis.
When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position. (The machine is in the in-position state.)

1827	In-position width in cutting feed for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
Set an in-position width for each axis in cutting feed. This parameter is used when bit 4 (CCI) of parameter No. 1801=1.

1828	Positioning deviation limit for each axis in movement

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
Set the positioning deviation limit in movement for each axis.

If the positioning deviation exceeds the positioning deviation limit during movement, a servo alarm SV0411, “EXCESS ERROR (MOVING)” is generated, and operation is stopped immediately (as in emergency stop).
Generally, set the positioning deviation for rapid traverse plus some margin in this parameter. Refer to the following set value.

$$\textbf{Setting value} = \frac{\textbf{Rapid traverse rate}^{(\text{Note 1})}}{\textbf{60} \times \textbf{Servo loop gain}^{(\text{Note 2})}} \times \frac{1}{\textbf{Detection unit}} \times 1.2 \textbf{ to } 1.5$$

⚠ CAUTION
If this parameter is not set correctly, machine or workpiece can be damaged.

NOTE
1 Usually, this value is parameter No.1420. When the maximal feedrate of each axis exceeds parameter No.1420 according to the command and override, the value is the maximal feedrate of the each axis.
2 Usually, this value is parameter No.1825. When the servo loop gains other than parameter No.1825 is effective, the value is actual servo loop gains.

1829	Positioning deviation limit for each axis in the stopped state

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in the stopped state for each axis.
If, in the stopped state, the positioning deviation exceeds the positioning deviation limit set for stopped state, a servo alarm SV0410, "EXCESS ERROR (STOP)" is generated, and operation is stopped immediately (as in emergency stop).

1850	Grid shift and reference position shift for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -99999999 to 99999999

To shift the reference position, the grid can be shifted by the amount set in this parameter.
Up to the maximum value counted by the reference counter can be specified as the grid shift.
In case of bit 4 (SFDx) of parameter No. 1008 is 0: Grid shift
In case of bit 4 (SFDx) of parameter No. 1008 is 1: Reference point shift

NOTE
For setting the reference position without dogs, only the grid shift function can be used.
(The reference position shift function cannot be used.)

1851	Backlash compensating value for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -9999 to 9999

Set the backlash compensating value for each axis.
When the machine moves in a direction opposite to the reference position return direction after the power is turned on, the first backlash compensation is performed.

3720	Number of position coder pulses

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Number of pulses
[Valid data range] Serial spindle

: 0, 4096 When bit 7 (FBP) of parameter No.3716 is set to 1: 4096×2^{14}

Spindle control with servo motor

: 0, 4096 When bit 7 (FBP) of parameter No.3716 is set to 1: 4096×2^{14}

Analog spindle

: 0, 4096 When bit 7 (FBP) of parameter No.3716 is set to 1: from 0 to 400000

Set the number of position coder pulses.
In analog spindle, 4096 is set to parameter automatically if parameter No.3720 is set to 0 or less.
In serial spindle or spindle control with servo motor, the setting method is changed by setting of the bit 7 (FBP) of parameter No.3716.

- When the bit 7 (FBP) of parameter No.3716 is set to 0:
  4096 is set to parameter automatically if parameter No.3720 is set to 0 or less.
- When the bit 7 (FBP) of parameter No.3716 is set to 1:
  4096×2^{14} is set to parameter automatically.

3721	Number of gear teeth on the position coder side

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 9999

Set the number of gear teeth on the position coder side in speed control (such as feed per revolution).

3722	Number of gear teeth on the spindle side

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 9999

Set the number of gear teeth on the spindle side in speed control (such as feed per revolution).

⚠ **CAUTION**

1. Parameter No.3721 and No.3722 can not be used for thread cutting.
   If thread cutting was done by setting the any value on these parameters, the correct screw can't be cut.
   In the machine to do thread cutting, connect mechanically between the spindle and position coder by the gear ratio=1:1. And these parameters set the No.3721=0, No.3722=0.
   (It is the setting for the gear ratio 1:1.)
2. Parameter No.3721 and No.3722 can not be used for Cs contour control. If Cs contour control was done by setting the any value on these parameters, spindle can't move as commanded. In the machine to do Cs contour control, connect mechanically between the spindle and position coder by the gear ratio = 1:1. And these parameters set the No.3721=0, No.3722=0. (It is the setting for the gear ratio 1:1.)

	#7	#6	#5	#4	#3	#2	#1	#0
4000				RETSV				

[Input type] Parameter input
[Data type] Bit spindle

#4 RETSV Reference position return direction of spindle on servo mode. (spindle positioning, rigid tapping, etc.)
0 : CCW (Counter clockwise)
1 : CW (Clockwise)

NOTE
The direction for spindle orientation (or reference position return) in spindle positioning using a serial spindle is determined by this parameter.

4056	Gear ratio (HIGH)
4057	Gear ration (MEDIUM HIGH)
4058	Gear ratio (MEDIUM LOW)
4059	Gear ratio (LOW)

[Data type] Word spindle
[Unit of data] Motor speed per spindle rotation × 100
[Valid data range] 0 to 32767
These parameters set the gear ration between the spindle and spindle motor.

NOTE
Set the gear ration between spindle and AC spindle motor when the spindle positioning is performed with serial spindle. For which gear is used, it depends on the clutch/gear signal (serial spindle) CTH1s, CTH1s.

4065	Position gain in servo mode (HIGH)
4066	Position gain in servo mode (MEDIUM HIGH)
4067	Position gain in servo mode (MEDIUM LOW)
4068	Position gain in servo mode (LOW)

[Data type] Word spindle
[Unit of data] 0.01 sec^{-1}
[Valid data range] 0 to 32767

These parameters set a servo loop gain on servo mode. (spindle positioning, rigid tapping, etc.)

NOTE
When the spindle positioning by a serial spindle is performed, set the position control loop gain in place of parameter No. 4970. For which gear is used, it depends on the clutch/gear signal (serial spindle) CTH1s, CTH1s.

4073	Grid shift amount on servo mode

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] 0to4096

This parameter sets a grid shift amount (distance from the position of a one-rotation signal to the machine zero point) in a servo mode (such as spindle positioning or rigid tapping).

NOTE
The setting of this parameter is used as a grid shift amount for orientation (reference position return) in spindle positioning using a serial spindle.

4074	Reference position return speed on servo mode

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] min^{-1}

The spindle speed set in this parameter is used for spindle orientation in a servo mode (such as spindle positioning or rigid tapping).

NOTE
The setting of this parameter is used as a grid shift amount for orientation (reference position return) in spindle positioning using a serial spindle.

	#7	#6	#5	#4	#3	#2	#1	#0
4950	IMBs	ESIs	TRVs			ISZs	IDMs	IORs

[Input type] Parameter input
[Data type] Bit spindle

#0 IORs Resetting the system in the spindle positioning mode
0: Does not release the mode.
1: Releases the mode

#1 IDMs The direction of spindle positioning (half-fixed angle positioning based on M codes) is:
0: Plus direction.
1: Minus direction.

#2 ISZs When an M code for switching to the spindle positioning mode is specified for spindle positioning:
0: The spindle is switched to the spindle positioning mode, and spindle orientation operation is performed.
1: Only the switching of the spindle to the spindle positioning mode is performed. (Spindle orientation operation is not performed.)

#5 TRVs The rotation direction for spindle positioning is:
0: Same as the specified sign.
1: Opposite to the specified sign.

> **NOTE**
> When a serial spindle is used, this parameter is invalid for the specification of a rotation direction for the orientation command.

#6 ESIs The unit of rapid traverse rate on the spindle positioning axis is:
0: Not increased by a factor of 10.
1: Increased by a factor of 10.

#7 IMBs When the spindle positioning function is used, half-fixed angle positioning based on M codes uses:
0: Specification A
1: Specification B

In the case of half-fixed angle positioning based on M codes, three types of spindle positioning operations can occur:
(1) The spindle rotation mode is cleared, then the mode is switched to the spindle positioning mode. (After switching to the spindle positioning mode, spindle orientation operation is also performed.)
(2) Spindle positioning is performed in the spindle positioning mode.
(3) The spindle positioning mode is cleared, then the mode is switched to the spindle rotation mode.

- In the case of specification A:
  Operations (1) to (3) are specified using separate M codes.
  (1) Specified using an M code for switching to the spindle positioning mode. (See parameter No. 4960)
  (2) Specified using M codes for specifying a spindle positioning angle. (See parameter No. 4962)
  (3) Specified using M codes for clearing spindle positioning operation. (See parameter No. 4961.)

- In the case of specification B:
  When M codes for specifying a spindle positioning angle are specified, operations (1) to (3) are performed successively. (See parameter No. 4962.) (However, spindle orientation operation of (1) is not performed.)

4960	M code specifying the spindle orientation

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 6 to 97
Set an M code for switching to the spindle positioning mode.

NOTE
1 Do not set an M code that duplicates other M codes used for spindle positioning.
2 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

4961	M code releasing the spindle positioning mode

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 6 to 97
Set an M code for canceling the spindle positioning mode on the spindle positioning axis.

NOTE
1 Do not set an M code that duplicates other M codes used for spindle positioning.
2 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

4962	M code for specifying a spindle positioning angle

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 6 to 9999999
Two methods are available for specifying spindle positioning. One method uses axis address for arbitrary-angle positioning. The other use an M code for half-fixed angle positioning. This parameter sets an M code for the latter method.
In this parameter, set an M code to be used for half-fixed angle positioning based on M codes.
Six M code from Mα to M(α+5) are used for half-fixed angle positioning, when α is the value of this parameter.

- When the number of M codes is set in parameter No. 4964, let α be the value set in parameter No. 4962, and let β be the value set in parameter No. 4964. Then, β M codes from Mα to M(α+β-1) are used as M codes for half-fixed angle positioning based on M codes.

The table below indicates the relationship between the M codes and positioning angles.

M code	Positioning angle	Example: Positioning angle when θ = 30°
Mα	θ	30°
M(α+1)	2θ	60°
M(α+2)	3θ	90°
M(α+3)	4θ	120°
M(α+4)	5θ	150°
M(α+5)	6θ	180°
:	:	:
M(α+β-1)	β×θ	β×30°

β represents the number of M codes set in parameter No. 4964.
(When parameter No. 4964 is set to 0, β = 6.)
θ represents the basic angular displacement set in parameter No. 4963.

NOTE
1. Do not set an M code that duplicates other M codes used for spindle positioning.
2. Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

4963	Basic angle for half-fixed angle positioning

[Input type] Parameter input
[Data type] Real spindle
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 to 60

This parameter sets a basic angular displacement used for half-fixed angle positioning using M codes.

4964	Number of M codes for specifying a spindle positioning angle

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 0 to 255

This parameter sets the number of M codes used for Half-fixed angle positioning using M codes.
As many M codes as the number specified in this parameter, starting with the M code specified in parameter No. 4962, are used to specify half-fixed angle positioning.
Let α be the value of parameter No. 4962, and let β be the value of parameter No. 4964. That is, M codes from Mα to M(α+β-1) are used for half-fixed angle positioning.
Setting this parameter to 0 has the same effect as setting 6. That is, M code from Mα to M(α+5) are used for half-fixed angle positioning.

NOTE
1. Make sure that M codes from Mα to M (α+β-1) do not duplicate other M codes.
2. Do not set an M code that duplicates other M codes used for spindle positioning.
3. Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

4970	Position gain

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 0.01/sec
[Valid data range] 1 to 9999

Set the position gain of the analog spindle in the spindle positioning mode.

4971 to 4974	Position gain multiplier (first stage) to Position gain multiplier (fourth stage)

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 1 to 32767

Set a position gain multiplier for an analog spindle in spindle positioning.
Position gain multiplier GC is obtained from the following equation:

$$GC = \frac{2048000 \times 360 \times PC \times E}{PLS \times SP \times L}$$

PLS Number of pulses output from the position coder (pulses/rev)
SP Number of gear teeth on the spindle side
PC Number of gear teeth on the position coder side
E Specified voltage (V) for turning the spindle motor at 1000 min^{-1}
L Angular displacement of the spindle (degrees) per spindle motor rotation

[Example] For the spindle motor and gear ratio given below, GC is calculated as follows:

PLS = 4096 pulse/rev
SP = 1
PC = 1
E = 2.2 V
L = 360 deg

$$GC = \frac{2048000 \times 360 \times 1 \times 2.2}{4096 \times 1 \times 360} = 1100$$

> **NOTE**
> On the assumption that the spindle motor used turns at 4500 min^{-1} at 10 V, 2.2 V is required to turn the spindle motor at 1000 min^{-1}

	#7	#6	#5	#4	#3	#2	#1	#0
11802				KSV				

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#4 KSV Servo axis is:
0: Enabled.
1: Disabled.

NOTE
1. This setting is effective regardless of the value of parameter No. 1023.
2. If this setting is made for the axis subject to Cs axis contour/spindle positioning, Cs axis contour/spindle positioning will be disabled.

Alarm and message

Number	Message	Description
PS0136	SPOS AXIS - OTHER AXIS SAME TIME	The spindle positioning axis and another axis are specified in the same block.
PS0137	M-CODE & MOVE CMD IN SAME BLK.	The spindle positioning axis and another axis are specified in the same block.
PS0194	SPINDLE COMMAND IN SYNCHRO-MODE	A Cs contour control mode, spindle positioning command, or rigid tapping mode was specified during the spindle synchronous control mode or spindle command synchronous control mode.
PS0224	ZERO RETURN NOT FINISHED	Reference position return has not been performed before the automatic operation starts. Perform reference position return only when the bit 0 (ZRNx) of parameter No. 1005 is set to 0. Or, spindle positioning command is specified in spindle rotation mode. Switch to spindle positioning mode and specify spindle positioning command.
PS1509	DUPLICATE M-CODE (SPOS AXIS ORIENTATION)	A function to which the same code as this M code is set exists. (spindle positioning, orientation)
PS1510	DUPLICATE M-CODE (SPOS AXIS POSITIONING)	A function to which the same code as this M code is set exists. (spindle positioning, positioning)
PS1511	DUPLICATE M-CODE (SPOS AXIS RELEASE)	A function to which the same code as this M code is set exists. (spindle positioning, mode cancel)
PS1543	ILLEGAL GEAR SETTING	The gear ratio between the spindle and position coder, or the set position coder number of pulses is illegal in the spindle positioning function.
SP0752	SPINDLE MODE CHANGE ERROR	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contour control, spindle positioning, rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.
SP1224	ILLEGAL SPINDLE-POSITION CODER GEAR RATIO	The spindle–position coder gear ratio was incorrect.
SP1233	POSITION CODER OVERFLOW	The error counter/speed instruction value of the position coder overflowed.
SP1234	GRID SHIFT OVERFLOW	Grid shift overflowed.
SP1240	DISCONNECT POSITION CODER	The analog spindle position coder is broken.
SP1243	ILLEGAL SPINDLE PARAMETER SETTING(GAIN)	The setting for the spindle position gain is incorrect.
SP1244	MOTION VALUE OVERFLOW	The amount of distribution to a spindle is too much
SV0410	EXCESS ERROR (STOP)	The amount of positional deviation during stopping exceeded the parameter No. 1829 setting value.
SV0411	EXCESS ERROR (MOVING)	The amount of positional deviation during traveling became excessive than the parameter setting value.

Number	Message	Description
SV1026	ILLEGAL AXIS ARRANGE	The parameter for servo axis arrange is not set correctly. - Parameter No. 1023 (servo axis number of each axis) is set to a negative value or a duplicate value. - The settings for parameter No. 1023 (servo axis number of each axis) were made with a certain setting skipped among 1 to 6, 9 to 14, or 17 to 22. - A setting of a multiple of 8 or a multiple of 8 minus 1 was made.

Caution

⚠ **CAUTION**

1. While the operations for spindle positioning (canceling the spindle rotation mode to enter the spindle positioning mode, positioning the spindle in the spindle positioning mode, and canceling the spindle positioning mode to enter the spindle rotation mode) are being performed, automatic operation stop signal *SP is invalid. This means that even when the * SP signal is set to 0, automatic operation does not stop until the entire sequence is completed.
2. Dry run and machine lock are not enabled during spindle positioning.
3. For M codes used for the spindle positioning function, auxiliary function lock is not enabled.
4. The spindle positioning function and the serial spindle Cs contour control function cannot be used together. If both functions are available (AXC(No.8133#1)=1 and SCS(No.8133#2)=1), positioning has priority.
5. The spindle positioning axis is treated as one of the controlled axes. Therefore, signals related to controlled axes (such as an overtravel signal) must be set.
6. When analog spindles are used, set exact drift compensation values.
   If an inappropriate drift compensation value is set, the positional deviation amount in the stopped state does not become 0. As a result, the following problems may occur when spindle orientation or spindle positioning is performed:
   - In-position wait state is entered, which results in a stop.
   - The spindle position is deviated. (The spindle does not reach or passes a specified position.)

- Note on using the rigid tapping function together

⚠ **CAUTION**

When using the rigid tapping function together with the spindle positioning function, do not specify rigid tapping in the spindle positioning mode, and contrariwise, do not specify spindle positioning in the rigid tapping mode.

Note

NOTE

1 M code commands related to spindle positioning must each be specified in an independent block. A block for specifying such an M code must not contain any other command. (An M command related to spindle positioning for another spindle must not be contained in the same block, either.)
Even when the function for enabling specification of multiple M codes in a single block is used together, spindle positioning M codes must be specified in an independent block.
2 The axis address for spindle positioning must be specified in an independent block. A block for specifying such an address must not contain any other command. The following commands, however, can be specified together with the axis address in the same block:
G00,G90,G91,G92 (M series, G code system B or C for T series)
G00,G50 (G code system A for T series)
3 M code commands related to spindle positioning are not buffered.
4 Spindle positioning cannot be performed by manual operations (such as a jog feed, manual handle feed, and so forth).
5 Spindle positioning cannot be performed through PMC axis control.
6 Program restart/block restart cannot be performed for spindle positioning. Use MDI for specification.
7 The stored stroke limit check is disabled for the spindle positioning axis.
8 The axis removal function is disabled for the spindle positioning axis.
9 The pitch error compensation function is disabled for the spindle positioning axis.
10 When a setting is made to omit spindle orientation, the reference position return completion signal is not set to 1.
11 In spindle orientation, all-axes interlock/each-axis interlock checks are made only at the start of the block. Even when the signal is input in the middle of block execution, it is ignored.
12 The difference between a move command and the actual movement amount is maintained until the spindle positioning mode is canceled.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	Auxiliary function/2nd auxiliary function
	Spindle serial output/analog output
	Spindle speed control

FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)

10.11 Cs CONTOUR CONTROL

10.11.1 Cs Contour Control

Overview

The Cs contour control function positions the serial spindle using the spindle motor in conjunction with a dedicated detector mounted on the spindle.
This function can perform more accurate positioning than the spindle positioning function, and has an interpolation capability with other servo axes.

- Increment system

As the least input increment of the Cs contour control axis, IS-B or IS-C must be set.
Selecting IS-C requires the following parameter settings:
Bit 1 (ISC) of parameter No. 1013 = 1
Bit 7 (CSC) of parameter No. 3729 = 1
Bit 0 (CS360M) of parameter No. 4005 = 1

- Command multiplication

Command multiplication for the Cs contour control axis must be set to 2 (1×).

Explanation

In Cs contour control, positioning is possible, as well as interpolation between the Cs contour control axis and a servo axis.

Performing speed control on a serial spindle is referred to as spindle rotation control (the spindle being rotated with a speed command), while performing position control on a spindle is referred to as spindle contour control (the spindle being rotated with a move command). The function for performing such spindle contour control is the Cs contour control function.
Switching between spindle speed control and CS contour control is performed by the DI signal from the PMC.
In the Cs contour control mode, the Cs contour control axis can be operated either manually or automatically, in the same way as normal servo axes.
(For a reference position return, see the relevant description in this section.)

- Setting the Cs contour control axis

If bit 7 (CSS) of parameter No. 3704 is 0:
Cs contour control is enabled for the first spindle in the path.
The axis used for Cs contour control must be set as an axis of the axes controlled by the CNC. Any controlled axis may be set as the Cs contour control axis.

Using parameter No. 1023, assign "-1" in the field corresponding to the chosen servo axis. (For single-path systems)
As the servo axis number (parameter No. 1023) of the axis used as the Cs contour control axis, set the number of the target logical spindle for Cs contour control with a minus sign (-1 to -n). (For multi-path systems)
This setting must be made only once for each control system. The spindle operating as the Cs contour control axis is the first spindle in the path.
If bit 7 (CSS) of parameter No. 3704 is 1:
Cs contour control can be performed for each spindle.
As the servo axis number (parameter No. 1023) of the axis used as the Cs contour control axis, set the number of the target logical spindle for Cs contour control with a minus sign (-1 to -n).
Cs contour control can be performed for more than one spindle at the same time.

The axis used as the Cs contour control axis must be set as a rotation axis (bit 0 (ROTx) of parameter No. 1006, and parameter No. 1022)

> ⚠ **CAUTION**
> Parameter No.3721 and No.3722 can not be used for Cs contour control. If Cs contour control was done by setting the any value on these parameters, spindle can't move as commanded. In the machine to do Cs contour control, connect mechanically between the spindle and position coder by the gear ratio = 1:1. And these parameters set the No.3721=0, No.3722=0. (It is the setting for the gear ratio 1:1.)

> **NOTE**
> If bit 4 (KSV) of parameter No.11802 is set to 1, Cs contour control will be disabled.

- Command Address

The address for the move command in Cs contour control is the axis name specified in parameter No. 1020. This address is arbitrary.

- Switching between spindle speed control and Cs contour control

- Switching from spindle speed control to Cs contour control
  If bit 7 (CSS) of parameter No. 3704 is 0:
    When the Cs contour control change signal CON <Gn027.7> is 1, the first spindle is regarded as the Cs contour control axis.
  If bit 7 (CSS) of parameter No. 3704 is 1:
    When the Cs contour control change signal for each spindle CONS1 to CONS4 <Gn274.0 to 3> is 1, the corresponding spindle is regarded as the Cs contour control axis.

When switching to the CS contour control mode is performed while the spindle is rotating, the spindle stops immediately to perform the change.

- Switching from Cs contour control mode to spindle speed control mode
  If bit 7 (CSS) of parameter No. 3704 is 0:
    When the Cs contour control change signal CON <Gn027.7> is 0, each spindle is placed in the spindle speed control mode.
  If bit 7 (CSS) of parameter No. 3704 is 1:
    When the Cs contour control change signal for each spindle CONS1 to CONS4 <Gn274.0 to 3> is 0, the corresponding spindle is placed in the spindle speed control mode.

Confirm that the move command for the spindle has been completed, then specify the switch. If it is switched while the spindle is moving, the machine enters interlock, or excessive position deviation alarm occurs.

- Setting axes for interpolation with a Cs contour control axis

Up to five servo axes can be used for interpolation with a Cs contour control axis. Any servo axes can be selected for interpolation with the Cs contour control axis by parameter setting. Set parameters as follows:

- When no servo axis is used for interpolation, specify 0 in parameters Nos. 3900, 3910, 3920, 3930, 3940.
- When one or more servo axes are used for interpolation, set the parameter for each as follows :

(1) Assign the axis number (1 to 8) to each of the servo axes used for interpolation in parameters Nos. 39n0 (n=0, 1, 2, 3, or 4).

(2) Set the loop gain for each of the servo axes specified in (1) in parameter nos. 39n1, 39n2, 39n3, 39n4. The loop gain must be the position loop gain for the Cs contour control axis or a desired value. Four parameters are provided to correspond to the four gears of the spindle. Use those parameters according to the inputs of the serial spindle clutch /gear signals (CTH1, CTH2).
(3) When the number of servo axes to be used for interpolation is smaller than five, set 0 in remaining parameter nos. 39n0.

Note on setting bit 7 (CSS) of parameter No. 3704 to 1
Common settings are used whichever spindle is used for Cs contour control. When Cs contour control is performed with more than one spindle, the parameters for the gear specified by the clutch/gear signals (CTH1 and CTH2) of the serial spindle for which the Cs contour mode is specified last are used.

When bit 7 (ALG) of parameter No. 1814 is set to 1, the loop gain can be set automatically at the time of switching between spindle speed control and Cs contour control.

- Automatic setting of the loop gain at the time of switching between spindle speed control and Cs contour control
  1. Switching from spindle speed control to Cs contour control
     The position loop gain of the Cs contour control axis selected by the clutch/gear signals (CTH1 and CTH2) (one of parameters Nos. 4069, 4070, 4071, and 4072) is set automatically for the servo axis for which bit 7 (ALG) of parameter No. 1814 is set to 1.
     When there are multiple Cs contour control axes, the smallest one of the position gain values selected for these axes is set automatically for the other Cs contour control axes and servo axes.
  2. Switching from Cs contour control to spindle speed control
     The original position gain (parameter No. 1825) is set automatically for the servo axis.
     When there are multiple Cs contour control axes, and there are still remaining Cs contour control axes, the smallest one of the position gain values selected for the individual remaining Cs contour control axes is set automatically for the other Cs contour control axes and servo axes.
  3. Switching between spindle speed control and Cs contour control during automatic operation
     Switching is impossible during automatic operation and during axis movement.
     If switching from spindle speed control to Cs contour control or vice versa is made in the middle of a block executed in automatic operation, position gains do not change immediately; they are set automatically after all controlled axes are stopped completely.
  4. Similarly, when switching between spindle speed control and Cs contour control is made in other than automatic operation, automatic setting is performed after the completely stopped states of all controlled axes are confirmed

- Reference Position Return of Cs Contour Control Axis

After the serial spindle is switched from spindle speed control to Cs contour control mode, the current position is undefined. Return the spindle to the reference position.
The reference position return of the Cs contour control axis is as follows:

- In manual mode
  After the serial spindle enters the Cs contour control mode, move the spindle in the direction of the reference position by turning on the feed axis and direction select signal (+Jn <Gn100> or -Jn <Gn102>). The spindle starts the movement to the reference position; when it reaches that position, the reference position return completion signal ZPn <Fn094> turns to 1. Turning any feed axis and direction select signal to 1 moves the tool in the reference position direction.
- In the automatic mode
  (a) G00 command
     If a serial spindle is placed in the Cs contour control mode, then the G00 command is issued without performing reference position return even once when bit 1 (NRF) of parameter No. 3700 is 0, the serial spindle performs a reference position return.
     As G00 is specified, the serial spindle indexes the reference position, then performs positioning at a specified position.

Only when positioning at the reference position is performed, the reference position return completion signal ZPn <Fn094> is set to 1 after completion of the positioning operation.
When a reference position return has been performed at least once, a G00 command performs a normal positioning operation.
When a reference position return is performed manually or by using G28, the spindle is always positioned at the reference position; on the other hand, when a reference position return is performed by using G00, the spindle can be positioned at any specified position.

> **NOTE**
> Set bit 0 (ZRN) of parameter No.1005 to 1.

(b) G28 command
After the serial spindle is put in the Cs contour control mode, issuing the G28 command stops the spindle motor, then moves the spindle to the midpoint. The spindle then returns to the reference position. At this point, the reference position return completion signal ZPn <Fn094> turns to 1.
When the serial spindle has returned to the reference position once while in the Cs contour control mode, the G28 command positions the spindle at the reference position without moving to the midpoint and ZPn comes on.

- Reference position return speed
After the serial spindle is put in the Cs contour control mode, the reference position return speed is the value of reference position return speed in Cs contour control (parameter No.4074). When reference position return speed in Cs contour control (parameter No.4074) is set to 0, the reference position return speed is the value of maximum speed in Cs contour control (parameter No.4021).
- Interruption of reference position return
(a) Manual operation
Return to the reference position can be interrupted by resetting, emergency stop, or turning off the feed axis and direction select signal. When the interrupted return operation is resumed, start from the beginning.
(b) Automatic operation
Return to the reference position can be interrupted by resetting, emergency stop, or feed hold. When the interrupted return operation is resumed, start from the beginning.

- Operation of Cs contour control axis (Manual/Automatic)

If a reference position return is performed on the Cs contour control axis, the axis can be operated in the same way as a normal NC axis.

In the spindle speed control mode, on the other hand, it does not operate as the Cs contour control axis, and alarm PS0197, "C-AXIS COMMANDED IN SPINDLE MODE" occurs during automatic operation.
In the spindle speed mode, inhibit manual operation of the Cs contour control axis using the PMC ladder.

Diagnosis data

- Display of Position Error of Cs Contour Control Axis

418	**Position deviation amount of each spindle**

Position deviation amount of the position loop for the each spindle.
This diagnostic data shows information obtained from the serial spindle control unit. This diagnosis data position error of the CS contour axis during CS contour control.
The position error can also be checked using a servo error display (DGN of No. 300) for an axis under Cs contour control.

300	**Position deviation amount of each servo axis**

Spindle tuning screen

In the Cs contour control mode, indications related to the Cs contour control mode appear.
For details, please refer to Spindle tuning screen of MAINTENANCE MANUAL (B-64695EN).

Signal

Cs contour control change signal CON<Gn027.7>

[Classification] Input signal
[Function] This signal specifies that the first spindle be switched between the spindle speed control and Cs contour control modes.
This signal is valid when bit 7 (CSS) of parameter No. 3704 is 0.

Cs contour control change signals in each axis
CONS1<Gn274.0> : First spindle
CONS2<Gn274.1> : Second spindle
CONS3<Gn274.2> : Third spindle
CONS4<Gn274.3> : Fourth spindle

[Classification] Input signal
[Function] This signal specifies that the first spindle be switched between the spindle speed control and Cs contour control modes.
This signal is valid when bit 7 (CSS) of parameter No. 3704 is 1. In this case, Cs contour control change signal CON <Gn027.7> is invalid.

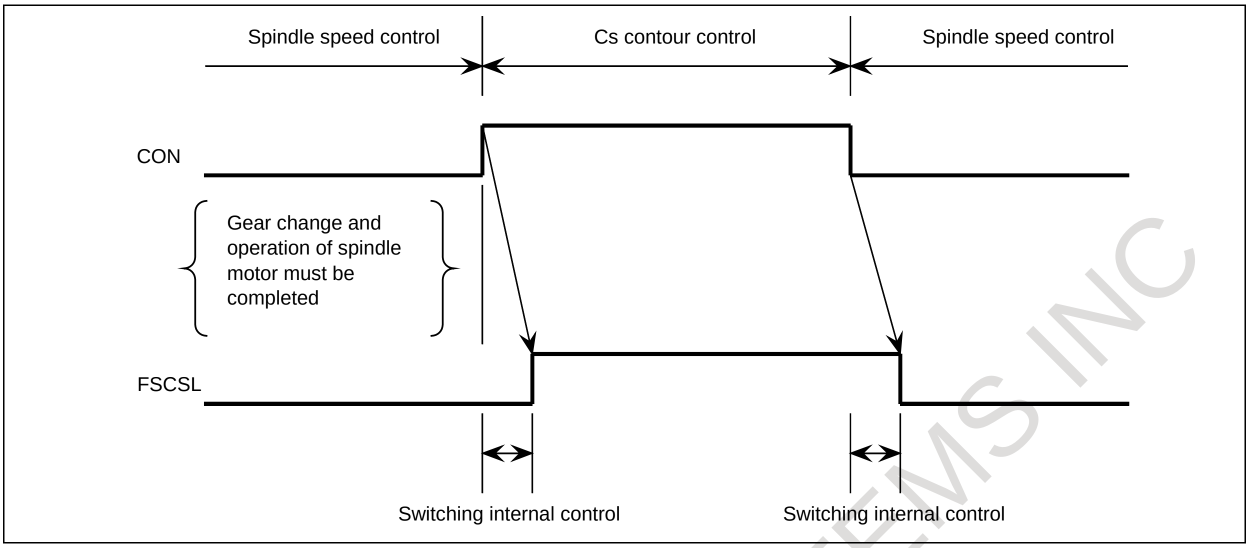
Cs contour control change completion signal FSCSL<Fn044.1>

[Classification] Output signal
[Function] This signal posts that the first spindle has switched to the Cs contour control mode.
[Output cond.] This signal is set to 1 when:
The spindle switches to the Cs contour control mode.
The signal is set to 0 when:
The spindle is not in the Cs contour control mode (is in the spindle speed control mode).
This signal is valid when bit 7 (CSS) of parameter No. 3704 is 0.

Cs contour control change completion signal in each axis
FCSS1<Fn274.0>:First spindle
FCSS2<Fn274.1>:Second spindle
FCSS3<Fn274.2>:Third spindle
FCSS4<Fn274.3>:Fourth spindle

[Classification] Output signal
[Function] This signal posts that the spindle has switched to the Cs contour control mode.
[Output cond.] This signal is set to 1 when:
The spindle switches to the Cs contour control mode.
The signal is set to 0 when:
The spindle is not in the Cs contour control mode (is in the spindle speed control mode).
This signal is valid when bit 7 (CSS) of parameter No. 3704 is 1. In this case, Cs contour control change completion signal FSCSL <Fn044.1> is invalid.

- Time Chart



NOTE
1. Any mechanical gear change needed and inputs for GR1, GR2, CTH1A, and CTH2A must be completed before the CON signal selects Cs contour control mode.
2. A servo excessive error may be generated if the spindle motor is not ready for operation. (Signal SRVA, SFRA <Gn070.4, 5> or other required signals must be appropriately processed on the machine side).
3. In the Cs contouring control mode, do not change the SFR (SRV) signal.

- Other signals

Gear select signals GR1O,GR2O,GR3O<Fn034.0 to 2>

[Classification] Output signal
[Function] This signal is output to the PMC to specify the gear stage.
[Output cond.] For information about this signal, see "SPINDLE SPEED CONTROL".

Gear select signals GR1,GR2<Gn028.1, Gn028.2>

[Classification] Input signal
[Function] This signal informs the CNC of the gear stage currently selected.
[Operation] For information about this signal, see "SPINDLE SPEED CONTROL".

Clutch/Gear signal (Serial spindle)
CTH1A,CTH2A<Gn070.3,Gn070.2> : First spindle
CTH1B,CTH2B<Gn074.3,Gn074.2> : Second spindle
CTH1C,CTH2C<Gn204.3,Gn204.2> : Third spindle
CTH1D,CTH2D<Gn266.3,Gn266.2> : Fourth spindle

Refer to the manual of serial spindle.

These signals determine what parameter (loop gain, etc.) to be used for each gear position.
CTH1 and CTH2 are the gear select signals for the serial spindle, but GR1 and GR2 must also be set. Do not change these signals while in the Cs contour control mode.

Relationship between gears selected and spindle gear select signals

CNC side							Serial spindle	
T/M series with CSSC			M series without CSSC					
GR2	GR1	Gear selection	GR3O	GR2O	GR1O	Gear selection	CTH1	CTH2
0	0	1st stage	0	0	0	1st stage	0	0
0	1	2nd stage	0	1	1	2nd stage	0	1
1	0	3rd stage	1	0	0	3rd stage	1	0
1	1	4th stage					1	1

CSSC: Constant surface speed control

NOTE
1. When the M series does not include the constant surface speed control (bit 0 (SSC) of parameter No. 8133=0), and bit 4 (GTT) of parameter No. 3706=0, GR1 and GR2 do not need to be input. Input CTH1 and CTH2 when gears are changed using GR1O, GR2O and GR3O that output from CNC.
2. The above combination of clutch/gear signals CTH1 and CTH2 is an example. The serial spindle gear is selected by CTH1 and CTH2 independently of gear selection on the CNC side. So, enter necessary signals, and set the corresponding serial spindle parameters.

Cs contour control axis reference position return completion signals ZP1 to ZP8<Fn094>

[Classification] Output signal

[Function] This signal indicates that a reference position return has been made for the Cs contour control axis.

ZPx

x : 1 ..... Reference position return completion signal of 1st spindle
2 ..... Reference position return completion signal of 2nd spindle
3 ..... Reference position return completion signal of 3rd spindle
: :
: :

[Output cond.] If a manual reference position return or automatic reference position return by G28 is performed during the Cs contour control mode, this signal becomes logical 1 when the Cs contour control axis reaches the reference position.

Signals on manual operation

Feed axis and direction select signal +Jn, -Jn <Gn100, Gn102> (Input) Manual handle feed axis select signal HSnA, HSnB, HSnC, HSnD <Gn018, Gn019> (Input) (Refer to respective items in this manual)

The Cs contour control axis can be manually operated in the same way as normal servo axes, except for a manual reference position return.

In the spindle speed control mode, however, manual operations for the Cs contour control axis must be inhibited using the PMC ladder, etc.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn027	CON							
Gn028						GR2	GR1	
Gn274					CONS4	CONS3	CONS2	CONS1
Fn034						GR3O	GR2O	GR1O
Fn044							FSCSL	
Fn094	ZP8	ZP7	ZP6	ZP5	ZP4	ZP3	ZP2	ZP1
Fn274					FCSS4	FCSS3	FCSS2	FCSS1

- For first serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn070	MRDYA		SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
Fn045			LDT2A	LDT1A				

- For second serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn074	MRDYB		SFRB	SRVB	CTH1B	CTH2B	TLMHB	TLMLB
Fn049			LDT2B	LDT1B				

- For third serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn204	MRDYC		SFRC	SRVC	CTH1C	CTH2C	TLMHC	TLMLC
Fn168			LDT2C	LDT1C				

- For fourth serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn266	MRDYD		SFRD	SRVD	CTH1D	CTH2D	TLMHD	TLMLD
Fn266			LDT2D	LDT1D				

Parameter

Major related parameters are described below.
In addition to the parameters described below, axis speed, acceleration, indication, and other parameters can be used. It is not necessary to set the digital servo parameters (parameters Nos. 2000 and up) for the axis used as the Cs contour control axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1005								ZRNx

[Input type] Parameter input
[Data type] Bit axis

#0 ZRNx If a move command other than G28 is specified by automatic operation when no reference position return is performed yet after the power is turned on:
0: The alarm PS0224, "ZERO RETURN NOT FINISHED." is issued.
1: Operation is performed without issuing an alarm.

> **NOTE**
> 1 The state in which a reference position has not been established refers to the following state:
> - When an absolute position detector is not used and reference position return has not been performed even once after power-up
> - When an absolute position detector is used and the association of the machine position with the position detected with the absolute position detector has not been completed (See the description of bit 4 (APZx) of parameter No. 1815.)
>
> 2 When the Cs axis coordinates are to be set up, set this parameter to 0.
> 3 To use a function that establishes the reference point and makes a movement with a command other than G28, such as an axis of Cs contour control, set this parameter to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1006					DIAx		ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx Setting linear or rotation axis.
#1 ROSx

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No. 1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.

ROSx	ROTx	Meaning
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

Set a Cs contouring control axis as a rotation axis.

> **NOTE**
> 1 Inch/metric conversion is not performed on rotation axes.
> 2 For rotation axes, the machine coordinate system is normalized to the range from 0 to 360 degrees. Automatic reference position return (G28, G30) is done in the direction of manual reference position return and the amount of travel does not exceed one turn.

#3 DIAx The move command for each axis is based on:
0: Radius specification
1: Diameter specification

For a Cs contouring control, set 0.

	#7	#6	#5	#4	#3	#2	#1	#0
1013							ISCx	ISAx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ISAx
#1 ISCx Increment system of each axis

Increment system	Bit 1 (ISC)	Bit 0 (ISA)
IS-A	0	1
IS-B	0	0
IS-C	1	0

1022	Setting of each axis in the basic coordinate system

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

To determine a plane for circular interpolation, cutter compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane), specify which of the basic three axes (X, Y, and Z) is used for each control axis, or a parallel axis of which basic axis is used for each control axis.
A basic axis (X, Y, or Z) can be specified only for one control axis.
Two or more control axes can be set as parallel axes for the same basic axis.

Setting	Meaning
0	Rotary axis (Neither the basic three axes nor a parallel axis)
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

In general, the increment system and diameter/radius specification of an axis set as a parallel axis are to be set in the same way as for the basic three axes.

For a Cs contouring control, set 0.

1023	Number of the servo axis for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 80
This parameter associates each control axis with a specific servo axis. Specify values 1+8n, 2+8n, 3+8n, 4+8n, 5+8n, and 6+8n (n = 0, 1, 2, …, 9) like 1, 2, 3, 4, 5, …, 77, and 78.
The control axis number is the order number that is used for setting the axis-type parameters or axis-type machine signals
With an axis for which Cs contour control/spindle positioning is to be performed, set -(spindle number) as the servo axis number.
Example)When exercising Cs contour control on the fourth controlled axis by using the first spindle, set -1.

1260	The shift amount per one rotation of a rotary axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the shift amount per one rotation of a rotary axis.

For the rotary axis used for cylindrical interpolation, set the standard value.

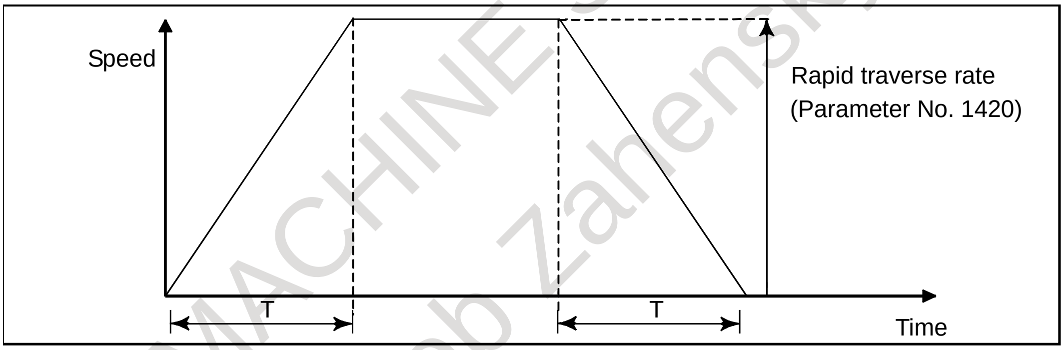
For a Cs contouring control, set 360.0.

1420	Rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

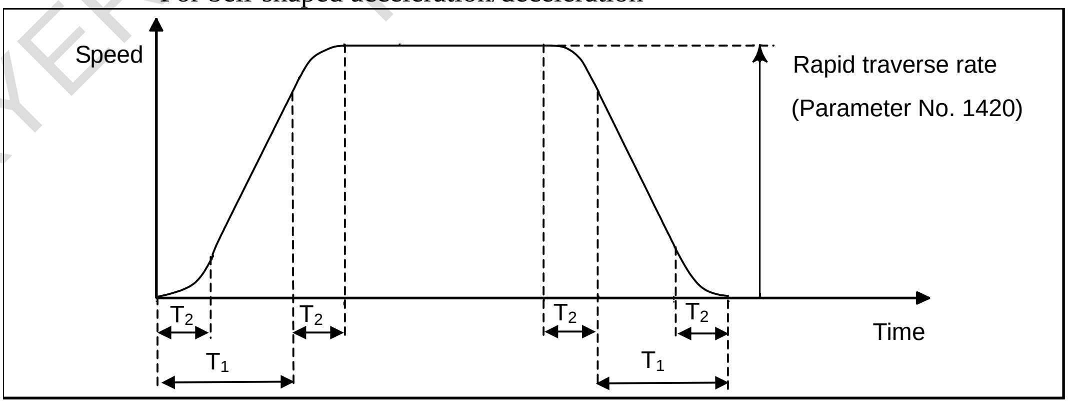
1620	Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
Specify a time constant used for acceleration/deceleration in rapid traverse.
[Example]
For linear acceleration/deceleration



T : Setting of parameter No. 1620

For bell-shaped acceleration/deceleration



T_1 : Setting of parameter No. 1620
T_2 : Setting of parameter No. 1621
(However, $T_1 \geq T_2$ must be satisfied.)
Total acceleration (deceleration) time : $T_1 + T_2$
Time for linear portion : $T_1 - T_2$
Time for curve portion : $T_2 \times 2$

1621	Time constant T_2 used for bell-shaped acceleration/deceleration in rapid traverse for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 512

Specify time constant T_2 used for bell-shaped acceleration/ deceleration in rapid traverse for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1814	ALGx							

[Input type] Parameter input
[Data type] Bit axis

#7 ALGx The servo axis loop gain in the Cs contour control mode is:
0: Not matched with the Cs contour control loop gain.
1: Matched with the Cs contour control loop gain.

NOTE
1 For those axes that are used for EGB, spindle positioning, index table indexing, PMC axis control, rigid tapping, and so forth, set this parameter to 0 (Not matched with the Cs contouring control loop gain).
2 When specifying axes subject to interpolation with the Cs contouring control axis by using parameter No. 39n0 (n = 0 to 4), set this parameter to 0.
3 When making a setting so that a position gain is automatically set at the time of Cs contouring control switching by setting bit 7 (ALG) of parameter No. 1814 to 1, set all of parameter No. 39n0 (n = 0 to 4) to 0.

1820	Command multiplier for each axis (CMR)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Unit of data] 0.5 times
[Valid data range] 1 to 96

For a Cs contouring control, set 2.

1825	Servo loop gain for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01/sec
[Valid data range] 1 to 32767

Set the loop gain for position control for each axis.

> **NOTE**
> If there is a servo axis that is subject to interpolation with the Cs contouring control axis, set parameters Nos. 3900 to 3944 to match the loop gain of the Cs contouring control axis to that of the servo axis in the Cs contour control mode.

1826	In-position width for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

The in-position width is set for each axis.
When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position. (The machine is in the in-position state.)

1828	Positioning deviation limit for each axis in movement

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.
If the positioning deviation exceeds the positioning deviation limit during movement, a servo alarm SV0411, "EXCESS ERROR (MOVING)" is generated, and operation is stopped immediately (as in emergency stop).
Generally, set the positioning deviation for rapid traverse plus some margin in this parameter. Refer to the following set value.

$$\textbf{Setting value} = \frac{\textbf{Rapid traverse rate}^{(\textbf{Note 1})}}{\textbf{60} \times \textbf{Servo loop gain}^{(\textbf{Note 2})}} \times \frac{1}{\textbf{Detection unit}} \times 1.2 \textbf{ to } 1.5$$

> ⚠ **CAUTION**
> If this parameter is not set correctly, machine or workpiece can be damaged.

> **NOTE**
> 1 Usually, this value is parameter No.1420. When the maximal feedrate of each axis exceeds parameter No.1420 according to the command and override, the value is the maximal feedrate of the each axis.
> 2 Usually, this value is parameter No.1825. When the servo loop gains other than parameter No.1825 is effective, the value is actual servo loop gains.

1829	Positioning deviation limit for each axis in the stopped state

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in the stopped state for each axis.

If, in the stopped state, the positioning deviation exceeds the positioning deviation limit set for stopped state, a servo alarm SV0410, "EXCESS ERROR (STOP)" is generated, and operation is stopped immediately (as in emergency stop).

1830	Axis-by-axis positional deviation limit at servo-off time

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

This parameter is used to set a positional deviation limit at servo-off time, on an axis-by-axis basis.
If the value specified with this parameter is exceeded at servo-off time, a servo alarm SV0410, "EXCESS ERROR (STOP)" is issued to cause an immediate stop (same as an emergency stop).
Usually, set the same value as a positional deviation at stop time.

> **NOTE**
> If this parameter is set to 0, the positioning deviation is not checked in servo-off state.

1851	Backlash compensating value for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -9999 to 9999

Set the backlash compensating value for each axis.
When the machine moves in a direction opposite to the reference position return direction after the power is turned on, the first backlash compensation is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
3700							NRF	

[Input type] Parameter input
[Data type] Bit path

#1 NRF With the first move command (G00) after switching the serial spindle to Cs contour control mode:
0: A reference position return operation is once performed then positioning is performed.
1: A normal positioning operation is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
3704	CSS							

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#7 CSS On the each spindle, Cs contour control is:
0: Not performed.
1: Performed.

	#7	#6	#5	#4	#3	#2	#1	#0
3716								A/Ss

[Input type] Parameter input
[Data type] Bit spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 A/Ss Spindle motor type is :
0: Analog spindle.
1: Serial spindle.

> **NOTE**
> When a serial spindle is used, enable the spindle serial output (bit 5 (SSN) of parameter No.8133is 0).

3717	Motor number to each spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled axes
Set a spindle amplifier number to be assigned to each spindle.
0: No spindle amplifier is connected.
1: Spindle motor connected to amplifier number 1 is used.
2: Spindle motor connected to amplifier number 2 is used.
to
n: Spindle motor connected to amplifier number n is used.

3721	Number of gear teeth on the position coder side

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 9999
Set the number of gear teeth on the position coder side in speed control (such as feed per revolution).

3722	Number of gear teeth on the spindle side

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 9999

Set the number of gear teeth on the spindle side in speed control (such as feed per revolution).

> ⚠ **CAUTION**
> 1 Parameter No.3721 and No.3722 can not be used for thread cutting. If thread cutting was done by setting the any value on these parameters, the correct screw can't be cut.
> In the machine to do thread cutting, connect mechanically between the spindle and position coder by the gear ratio=1:1. And these parameters set the No.3721=0, No.3722=0.
> (It is the setting for the gear ratio 1:1.)
> 2 Parameter No.3721 and No.3722 can not be used for Cs contour control. If Cs contour control was done by setting the any value on these parameters, spindle can't move as commanded. In the machine to do Cs contour control, connect mechanically between the spindle and position coder by the gear ratio = 1:1. And these parameters set the No.3721=0, No.3722=0. (It is the setting for the gear ratio 1:1.)

	#7	#6	#5	#4	#3	#2	#1	#0
3729	CSCs				NCSs	CSNs		

[Input type] Parameter input
[Data type] Bit spindle

#2 CSNs When the Cs contour control mode is turned off, an in-position check is:
0: Performed.
1: Not performed.

> **NOTE**
> This parameter is Automatic setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

#3 NCSs When the Cs contour control mode is set:
0: Switching to Cs contour control is completed when the spindle activating current is on (the spindle amplifier is ready for operation in the Cs contour control mode).
1: Switching to Cs contour control is completed even when the spindle activating current is off (the spindle amplifier is not ready for operation in the Cs contour control mode).

If this parameter is set to 1, the Cs contour control switch end signal is output without waiting for the spindle to decelerate to a stop.

#7 CSCs The increment system of the Cs contour control axis is:
0: IS-B.
1: IS-C.

3900	Number of servo axis for interpolation with Cs contour control axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to number of controlled axes

Set the number of the servo axis to be used for interpolation with the Cs contour control axis. (For group 1)

> **NOTE**
> When there is no servo axis used for interpolation with the Cs contour control axis, set 0.

3901	Loop gain for servo axis for interpolation with Cs contour control axis (HIGH gear)

3902	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM HIGH gear)

3903	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM LOW gear)

3904	Loop gain for servo axis for interpolation with Cs contour control axis (LOW gear)

[Input type] Parameter input
[Data type] Word path
[Unit of data] 0.01/sec
[Valid data range] 0 to 9999

Set the servo loop gain for each spindle gear of the servo axis used for interpolation with the Cs contour control axis. (For group 1)

3910	Number of servo axis for interpolation with Cs contour control axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to number of controlled axes

Set the number of the servo axis to be used for interpolation with the Cs contour control axis. (For group 2)

> **NOTE**
> When there is no servo axis for interpolation with the Cs contour control axis, or when there is not more than one servo axis, set 0.

3911	Loop gain for servo axis for interpolation with Cs contour control axis (HIGH gear)

3912	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM HIGH gear)

3913	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM LOW gear)

3914	Loop gain for servo axis for interpolation with Cs contour control axis (LOW gear)

[Input type] Parameter input
[Data type] Word path
[Unit of data] 0.01/sec
[Valid data range] 0 to 9999

Set the servo loop gain for each spindle gear of the servo axis used for interpolation with the Cs contour control axis. (For group 2)

3920	Number of servo axis for interpolation with Cs contour control axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to number of controlled axes

Set the number of the servo axis to be used for interpolation with the Cs contour control axis. (For group 3)

> **NOTE**
> When there is no servo axis for interpolation with the Cs contour control axis, or when there is not more than two servo axes, set 0.

3921	Loop gain for servo axis for interpolation with Cs contour control axis (HIGH gear)

3922	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM HIGH gear)

3923	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM LOW gear)

3924	Loop gain for servo axis for interpolation with Cs contour control axis (LOW gear)

[Input type] Parameter input
[Data type] Word path
[Unit of data] 0.01/sec
[Valid data range] 0 to 9999

Set the servo loop gain for each spindle gear of the servo axis used for interpolation with the Cs contour control axis. (For group 3)

3930	Number of servo axis for interpolation with Cs contour control axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to number of controlled axes

Set the number of the servo axis to be used for interpolation with the Cs contour control axis. (For group 4)

> **NOTE**
> When there is no servo axis for interpolation with the Cs contour control axis, or when there is not more than three servo axes, set 0.

3931	Loop gain for servo axis for interpolation with Cs contour control axis (HIGH gear)

3932	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM HIGH gear)

3933	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM LOW gear)

3934	Loop gain for servo axis for interpolation with Cs contour control axis (LOW gear)

[Input type] Parameter input
[Data type] Word path
[Unit of data] 0.01/sec
[Valid data range] 0 to 9999

Set the servo loop gain for each spindle gear of the servo axis used for interpolation with the Cs contour control axis. (For group 4)

3940	Number of servo axis for interpolation with Cs contour control axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to number of controlled axes

Set the number of the servo axis to be used for interpolation with the Cs contour control axis. (For group 5)

> **NOTE**
> When there is no servo axis for interpolation with the Cs contour control axis, or when there is not more than four servo axes, set 0.

3941	Loop gain for servo axis for interpolation with Cs contour control axis (HIGH gear)

3942	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM HIGH gear)

3943	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM LOW gear)

3944	Loop gain for servo axis for interpolation with Cs contour control axis (LOW gear)

[Input type] Parameter input
[Data type] Word path
[Unit of data] 0.01/sec
[Valid data range] 0 to 9999

Set the servo loop gain for each spindle gear of the servo axis used for interpolation with the Cs contour control axis. (For group 5)

	#7	#6	#5	#4	#3	#2	#1	#0
4000					CSO			

[Input type] Parameter input
[Data type] Bit spindle

#3 CSO Direction of reference position return in Cs contour control

0: Reference position return is performed by rotating the spindle counterclockwise (when viewed from the motor output axis).

1: Reference position return is performed by rotating the spindle clockwise (when viewed from the motor output axis).

4021	Maximum speed in Cs contour control

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] min^{-1}

Set the maximum speed of the spindle in Cs contour control.

4056	Gear ratio (HIGH gear)

4057	Gear ratio (MEDIUM HIGH gear)

4058	Gear ratio (MEDIUM LOW gear)

4059	Gear ratio (LOW gear)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Motor speed per spindle rotation × 100
[Valid data range] 0 to 32767

Set the gear ratio of the spindle to spindle motor.

> **NOTE**
> When a serial spindle is positioned, set the gear ratio of the spindle to spindle motor.
> The gear ratio to be used for spindle operations is set by clutch/gear signals (serial spindle) CTH1s and CTH2s.

4069	Position gain in Cs contour control (HIGH gear)

4070	Position gain in Cs contour control (MEDIUM HIGH gear)

4071	Position gain in Cs contour control (MEDIUM LOW gear)

4072	Position gain in Cs contour control (LOW gear)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 0.01/sec
[Valid data range] 0 to 32767

Set the position gain in Cs contour control.

4074	Reference position return speed in servo mode

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] min^{-1}

As the spindle orientation speed in a servo mode (spindle positioning, rigid tapping, and so on), the spindle speed set in this parameter is used.

> **NOTE**
> When a serial spindle is positioned, the orientation (reference position return) speed depends on this parameter.

4135	Grid shift amount in Cs contour control

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Detection unit
[Valid data range] Within ±1 rotation (For 1-micron detection, for example, -360000 to 360000)

Set the grid shift amount (the distance from the one-rotation signal position to machine zero point) in Cs contour control.

	#7	#6	#5	#4	#3	#2	#1	#0
11802				KSV				

[Input type] Parameter input

[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#4 KSV Servo axis is:
0: Enabled.
1: Disabled.

> **NOTE**
> 1 This setting is effective regardless of the value of parameter No. 1023.
> 2 If this setting is made for the axis subject to Cs axis contour/spindle positioning, Cs axis contour/spindle positioning will be disabled.

Alarm and message

Number	Message	Description
PS0194	SPINDLE COMMAND IN SYNCHRO-MODE	A Cs contour control mode, spindle positioning command, or rigid tapping mode was specified during the spindle synchronous control mode or spindle command synchronous control mode.
PS0197	C-AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs-axis when the Cs contour control switching signal was off.
SP0752	SPINDLE MODE CHANGE ERROR	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contour control, spindle positioning, rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.

Caution

> ⚠ **CAUTION**
> 1 The M codes to specify Cs contour control mode switching must be assigned to M codes not buffered. (Parameters Nos. 3411 to 3420 and 11290 to 11299)
> 2 Avoid changing spindle gears in Cs contour control mode. When a gear change is required, be sure to perform the gear change in spindle speed control mode.
> 3 The following error compensation functions are enabled for the Cs contour control axis:
> Pitch error compensation, both-direction pitch error compensation, backlash compensation

⚠ CAUTION

4 For the axis under Cs contour control, the following functions cannot be used:
- Index table indexing
- Axis synchronous control
- Axis switching
- PMC axis control (However, it is possible to use under Cs contour control when Rapid traverse (00h), Cutting feed (01h), Continuous feed (06h), Machine coordinate system selection (20h), Speed command (10h) and position control is selected.
(when bit 2(VCP) of parameter No.8007 is set to 1 and bit 1 (EVP) of No.8005is set to 1))
- Polar coordinate command
- Removal of controlled axis

5 In the Cs contour control mode, when an axis control parameter for the Cs contour control axis such as the acceleration/deceleration time constant is changed, the new parameter value does not become valid immediately. The new parameter value becomes valid after the Cs contour control mode is canceled then entered again.

6 When the position gain is automatically set at Cs contour control switching (when bit 7 (ALG) of parameter No. 1814 is set to 1), changing the setting in parameters Nos. 4069 to 4072 of the loop gain corresponding to the selected spindle gear does not make the new setting valid immediately.
To make the new setting valid, cancel the Cs contour control mode, then set the Cs contour control mode again. If more than one Cs contour control axis is present, however, the newly set value is included in the position gain values selected for these axes among which the smallest value is to be selected.

7 When the axis under Cs contour control is turned by speed command (10h) of PMC axis control, it is necessary to set parameter No.2023 (Number of velocity pulses(PULCO)).

Reference item

For details of the parameters (parameters Nos. 4000 to 4539), signals, and alarms for the serial spindle control unit, refer to the following manuals of serial spindles:
FANUC AC SPINDLE MOTOR α*i* series PARAMETER MANUAL (B-65280EN)

10.11.2 Cs Contour Control Torque Limit Skip

Overview

This function enables a torque limit skip for the Cs contour control axis.

Explanation

Torque limit command signal TLMH and load detection signal LDT1 for serial spindles are used to make a torque limit skip for the Cs contour control axis.
With this function, if load detection signal LDT1 of a serial spindle operated under Cs contour control is set to 1 when a torque limit skip is specified (G31 P98/P99), skip processing is performed.

To use this function, make the following settings:
(1) Set bit 0 (CST) of parameter No. 6215 for the corresponding Cs contour control axis to 1.
(2) Set a torque limit in parameter No. 4025. (Unit: %, maximum torque ratio)

(3) Set torque limit command signal TLMH to 1. In this signal state, the output torque is limited to the value set in parameter No. 4025.
(4) Set the load detection level in parameter No. 4026. This setting must be slightly smaller than the torque limit set in step (2) above.

When this load detection level is reached, load detection signal LDT1 is output, enabling skip processing if a torque limit skip is specified (G31 P98/P99).

The other specifications are the same as for normal torque limit skip. For details of the torque limit skip, refer to the description of torque limit skip in "Connection Manual (Function)" and the description of torque limit skip in "Operator’s Manual".
For torque limit command signal TLMH and load detection signal LDT1 for serial spindles, refer to relevant manuals such as "FANUC Spindle Motor αi series PARAMETER MANUAL" (B-65280EN).

NOTE

The torque limit override function by address Q (when bit 0 (TQO) of parameter No. 6286 is set to 1) is disabled for Cs contour control axes. PMC sequence programs must be created so that Cs contour control axes are always placed in the torque limit state before the G31 P99/P98 is specified.

(Sample program)

```
O0012
:
Mxx (Specify torque limit from PMC)
:
G31 P99 C200.F100. (Torque limit skip command)
:
Mxx (Cancel torque limit from PMC)
:
M30
```

Signal

Torque limit command signal (serial spindle)

TLMHA<Gn070.1> : First spindle
TLMHB<Gn074.1> : Second spindle
TLMHC<Gn204.1> : Third spindle
TLMHD<Gn266.1> : Fourth spindle

[Classification] Input signal
[Function] These signals limit the output torque of the corresponding serial spindle to the value set in parameter No. 4025. To cause a torque limit skip for a Cs contour control axis, set this signal to 1. For details, refer to the relevant manual of the serial spindle.

Load detection signal (serial spindle)

LDT1A <Fn045.4> : First spindle
LDT1B <Fn049.4> : Second spindle
LDT1C <Fn168.4> : Third spindle
LDT1D <Fn266.4> : Fourth spindle

[Classification] Output signal
[Function] These signals are output when the load on the corresponding serial spindle has reached the value set in parameter No. 4026.
For details, refer to the relevant manual of the serial spindle.

Torque limit state signal (serial spindle)
TLMA <Fn045.6> : First spindle
TLMB <Fn049.6> : Second spindle
TLMC <Fn168.6> : Third spindle
TLMD <Fn266.6> : Fourth spindle

[Classification] Output signal

[Function] These signals are output when the torque is being limited by the corresponding torque limit command signal.
For details, refer to the relevant manual on the serial spindle.

Signal address

- For first serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn070							TLMHA	
Fn045		TLMA		LDT1A				

- For second serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn074							TLMHB	
Fn049		TLMB		LDT1B				

- For third serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn204							TLMHC	
Fn168		TLMC		LDT1C				

- For fourth serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn266							TLMHD	
Fn266		TLMD		LDT1D				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6215								CSTx

[Input type] Parameter input
[Data type] Bit axis

#0 CSTx On a Cs contour control axis, torque limit skip operation is:
0: Not performed.
1: Performed.
Torque limit skip operation is performed using the torque limit command signal TLMH and the load detection signal LDT1 of the serial spindle.

	#7	#6	#5	#4	#3	#2	#1	#0
4009				LDTOUT				

[Input type] Parameter input
[Data type] Bit spindle

#4 LDTOUT Whether to output the load detection signals (LDT1 and LDT2) during acceleration/deceleration:
0: Not output during acceleration/deceleration. (standard setting value)
1: Output (at all times) during acceleration/deceleration if the parameter-set level is exceeded.

4025	Torque limit value

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%
[Valid data range] 0to100

Set a torque limit value applicable when a torque limit command (TLMH) is specified. Specify limit value data, assuming that the maximum torque represents 100%.

4026	Load detection level 1

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%
[Valid data range] 0 to 100

Set the detection range of the load detection signal (LDT1).
When the motor output exceeds (setting data)% of the maximum output, the load detection signal LDT1 is set to 1.

NOTE
1 If torque limit skip is specified (with bit 0 of parameter No. 6215 set to 1) for a Cs contouring control axis, an excessive error check during a stop is not made for the axis when the load detection state (LDT1 = 1) is set while the torque limit command is specified (TLMH = 1) in the Cs contouring control mode.
2 If torque limit skip is specified (with bit 0 of parameter No. 6215 set to 1) for a Cs contouring control axis, an in-position check is not made for the axis when the load detection state (LDT1 = 1) is set in the Cs contouring control mode.

10.11.3 Cs Contour Control Arbitrary Reference Position Setting Function

Overview

When bit 0 (CRF) of parameter No. 3700 is set to 1, any position can be set as the reference position by the first reference position return command (G28 or a manual reference position return) issued after the serial spindle enters the Cs contour control mode.

Explanation

When the first reference position return command (G28 or a manual reference position return) is executed after the serial spindle enters the Cs contour control mode, a reference position return is performed with the current position assumed to be the reference position. Therefore, a spindle movement for positioning at the reference position is not made.
Since the reference position return operation does not involve positioning to a spindle position specific to the spindle, the cycle time can be reduced.

NOTE

1 The spindle position at a reference position set by a reference position return operation by using this function is not guaranteed to match the spindle position at a reference position set by a reference position return operation performed again after the Cs contour control mode is turned off then back on.
2 When an intermediate point is specified in G28, a movement to the intermediate point is made, then that position is set as the reference position.
3 When this function is used, and a G00 command is issued for a Cs contour control axis without performing a reference position return even once after the serial spindle is placed in the Cs contour control mode, alarm PS0303, "REFERENCE POSITION RETURN IS NOT PERFORMED" is issued even if bit 1 (NRF) of parameter No. 3700 is set to 0. A reference position return must always be performed using the G28 command.
4 When this function is used, grid shift at the time of reference position return (parameter No. 4135) is invalid.
5 In automatic phase matching of the spindle EGB, if the master axis is a Cs contour control axis, this function cannot be used.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3700							NRF	CRF

[Input type] Parameter input
[Data type] Bit path

#0 CRF Reference position setting at an arbitrary position under Cs contour control is:
0: Not used.
1: Used.

#1 NRF With the first move command (G00) after switching the serial spindle to Cs contour control mode:
0: A reference position return operation is once performed then positioning is performed.
1: A normal positioning operation is performed.

NOTE
When this function is used, an attempt to specify G00 for a Cs contour control axis without performing a reference position return operation even once after switching the serial spindle to the Cs contour control mode results in the alarm PS0303, "REFERENCE POSITION RETURN IS NOT PERFORMED" even if bit 1 (NRF) of parameter No. 3700 is set to 0. Be sure to perform a reference position return operation by specifying G28.

4135	Grid shift amount on Cs contouring control

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] 1 pulse unit (=0.001°) (0.0001° when bit 0 (CS360M) of parameter No. 4005 is set to 1)
[Valid data range] -360000 to +360000
(-3,600,000 to +3,600,000 when bit 0 (CS360M) of parameter No. 4005 is set to 1)
Use this parameter to shift the machine reference position on Cs contouring control.

The machine reference position of the spindle shifts by the set number of pulses in the CCW direction.

Alarm and message

Number	Message	Description
PS0303	REFERENCE POSITION RETURN IS NOT PERFORMED	When the setting of a reference position at any position was possible in Cs contour control (bit 0 (CRF) of parameter No. 3700 = 1), a G00 command was issued for the Cs contour axis without a return to the reference position after the serial spindle was switched to Cs contour control mode. Perform a reference position return with a G28 command before issuing a G00 command.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	Cs contour control

10.11.4 Cs Contour Control Axis Coordinate Establishment

Overview

Shifting a serial spindle from spindle rotation control to Cs contour control results in its current position being lost.
This function is intended to establish the current position without making a reference position return. This is done by setting the Cs axis coordinate establishment request signal CSFIx<Gn274.4 to 7> to 1.

NOTE

Once the power has been turned on, this function remains enabled until the power is turned off after the return of the Cs contour axis to the reference position.
However, when you use αiCZ sensor (serial type) as a spindle sensor and the Z-phase signal is detected one or more times after power on, this function is available without reference position return.
Moreover, when you use a distance coded sensor and the Z-phase signal is detected three or more times after power on, this function is available without reference position return.
Moreover, when you use the serial type sensor (RCN2390F (made by HEIDENHAIN) etc.), which recognizes the position within one revolution just after power on, as a spindle sensor, this function is available without reference position return.

Explanation

- Setting

This function is enabled by setting bit 2 (CSF) of parameter No. 3712 and bit 5 (CSPTRE) of parameter No. 4353 to 1.
(Using this function requires resetting bit 7 (RFCHK3) of parameter No. 4016 to 0.)

Inhibit buffering with the M code (parameters Nos. 3411 to 3432 and 11290 to 11299) used to set the Cs axis coordinate establishment request signal CSFIx<Gn274.4 to 7> to 1.

- Cs axis coordinate establishment procedure

(1) Turn the Cs contour control mode ON, using the M code as a trigger, and set the Cs axis coordinate establishment request signal CSFIx<Gn274.4 to 7> to 1.
(2) The absolute, relative, and machine coordinate systems for the Cs axis are established if the Cs axis origin established state signal CSPENx is 1.
(Establishing the relative coordinate system requires setting bit 3 (PPD) of parameter No. 3104 to 1.)
(3) Once the coordinate systems are established, the reference position established signal ZRFx<Fn120.0 to 7> becomes 1. So, reset the Cs axis coordinate establishment request signal CSFIx<Gn274.4 to 7> to 0.
(4) Perform the FIN processing for the M code.

NOTE
1 When inputting the MRDY, SFR, and SRV signals from the PMC to serial spindles via the CNC, ensure they are input normally.
2 In an emergency stop or servo alarm condition, no Cs axis coordinate system can be established.
3 Do not shift the Cs contour axis when Cs axis coordinate establishment is under way.
4 Generally, when Cs contouring axis performs reference position return after power on, CSPENx becomes 1. However, CSPENx becomes 1 even without reference position return when the following sensors are used as spindle sensors.
- αiCZ sensor (serial type)
  When Z-phase signal is detected one or more times after power on, CSPENx becomes 1.
- Distance coded sensor
  When Z-phase signal is detected three or more time after power on, CSPENx becomes 1.
- The serial type sensor (RCN2390F (made by HEIDENHAIN) etc) which recognizes the position within one revolution just after power on, CSPENx becomes 1.

- Case in which no Cs axis coordinate establishment can be performed

If Cs axis coordinate establishment cannot be performed, alarm PS5346 is issued for the path where the absolute coordinates (workpiece coordinates) are is to be established.

The following conditions prevent Cs axis coordinate establishment.
- The Cs axis origin established state signal CSPENx is 0.
- An attempt to transfer positional information from the spindle amplifier has failed.
- The servo section is off when an attempt is made to start Cs axis coordinate establishment.
- The Cs axis is under synchronous control or superimposed control.
- An emergency stop condition has occurred during coordinate establishment.
- The Cs axis has exited composite control during coordinate establishment.
- An attempt has been made to start synchronous, composite, or superimposed control for the Cs axis during coordinate establishment.

The Cs axis coordinate establishment alarm signal CSFOx<Fn274.4 to 7> becomes 1 in the path to which the spindle is connected. So, reset the Cs axis coordinate establishment request signal CSFIx<Gn274.4 to 7> to 0.
Resetting the Cs axis coordinate establishment request signal CSFIx<Gn274.4 to 7> to 0 causes the Cs axis coordinate establishment alarm signal CSFOx<Fn274.4 to 7> to be reset to 0.

- Clearing alarm PS5346

Alarm PS5346 cannot be cleared before the Cs axis reference return position is established.
In the path where alarm PS5346 occurred, make a manual reference position return for the Cs axis.
Once the reference position has been established, a reset can clear alarm PS5346.

Performing a reset after the Cs contour control mode is exited enables alarm PS5346 to be cleared.

- Serial type sensor which recognizes the position within one revolution just after power on

When you use the serial type sensor which recognizes the position within one revolution just after power on, CSPENx becomes 1 without reference position return, and Cs Contour Control Axis Coordinate Establishment is available.

The example of applicable sensor is listed below.

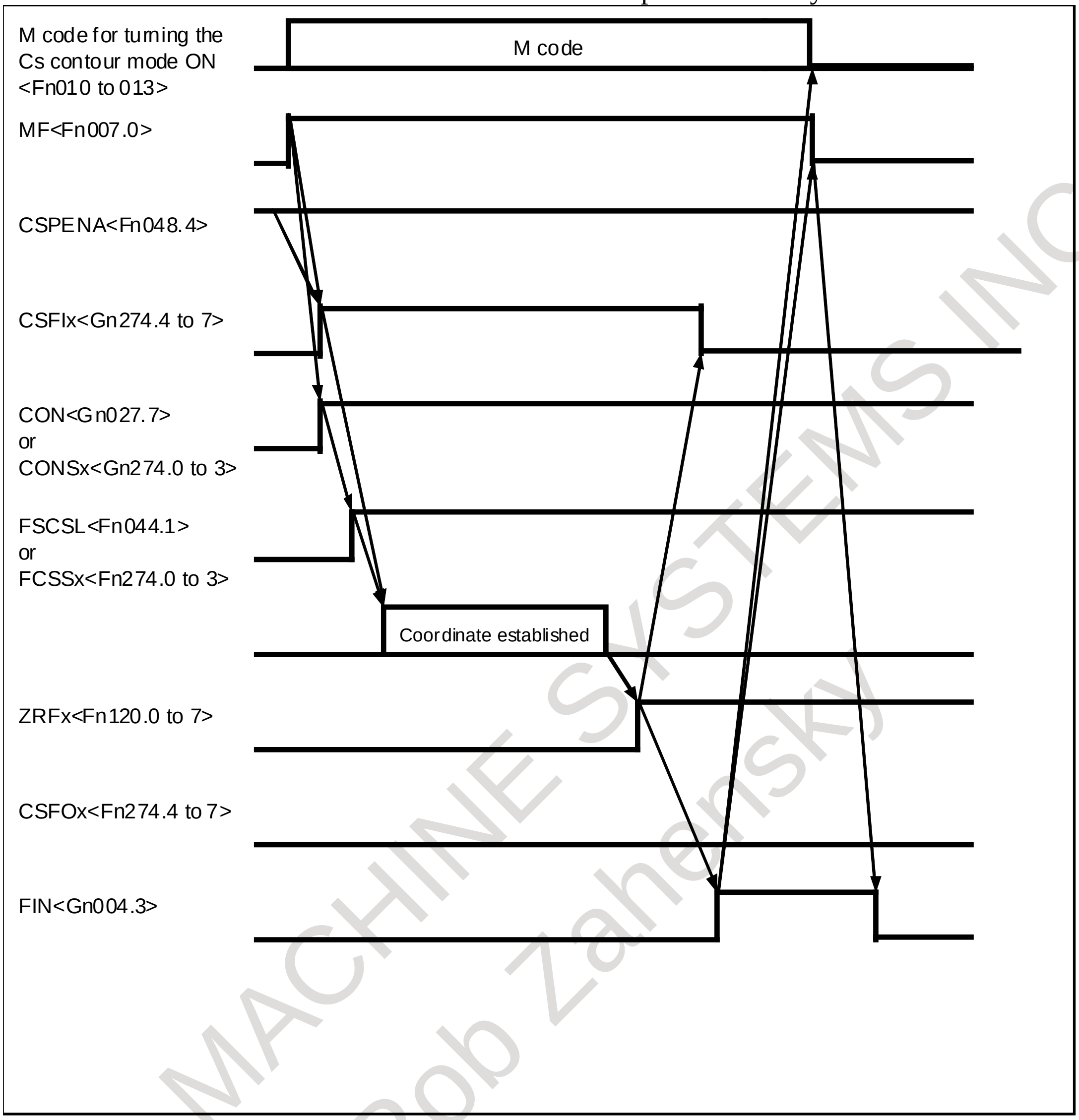
- HEIDENHAIN
  RCN2390F
  RCN2590F
  RCN5390F
  RCN5590F
  RCN8390F
  RCN8590F

NOTE

1 To use the above sensor, the corresponding spindle software is required.
2 To use the above sensor as a spindle sensor, the following parameters should be set.
   - No.4002(#3,#2,#1,#0)=(0, 1, 1, 0)
   - No.4361=8192
   - No.4548#0=1

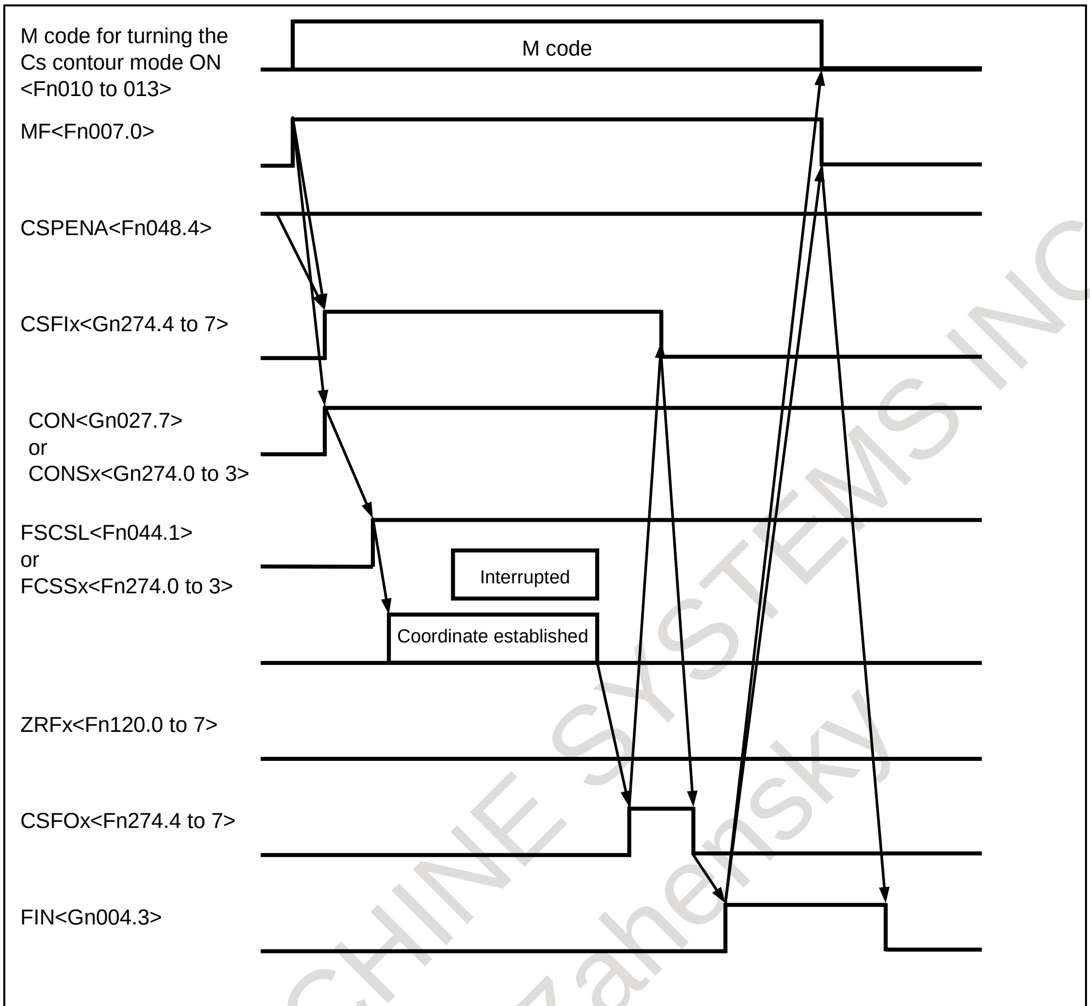
- Sequence (timing chart)

Case in which Cs axis coordinate establishment has been completed normally



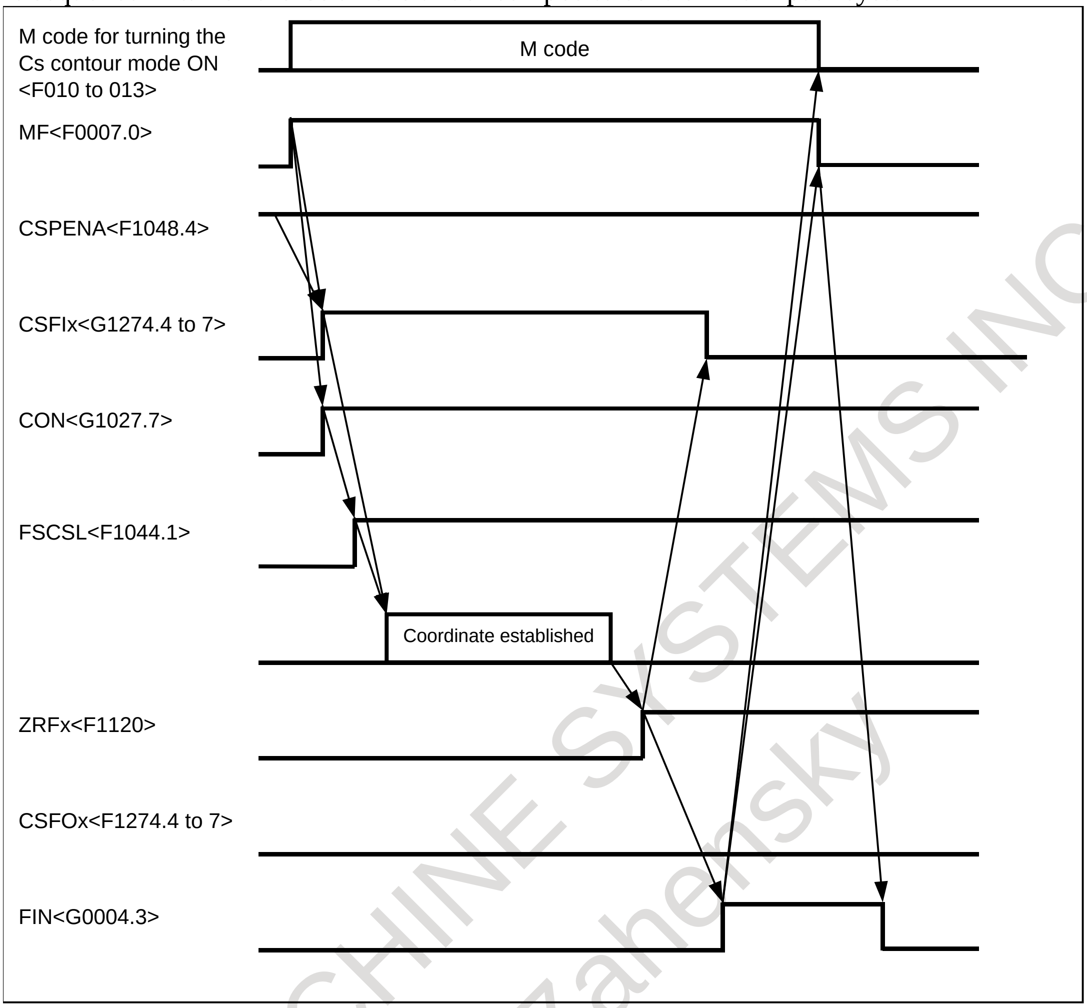
Cs axis coordinate establishment remains enabled in the Cs contour mode.
In any mode other than the Cs contour mode, setting the Cs axis coordinate establishment request signal CSFIx<Gn274.4 to 7> to 1 causes Cs axis coordinate establishment to be ignored.

Case in which Cs axis coordinate establishment cannot be achieved



A failure in coordinate establishment prevents the ZRFx signal <Fn120.0 to 7> from being set to "1".

Example sequence in which the Cs axis is under composite control in a 2-path system



(1) In path 1, issuing an M code causes the MF signal <F0007.0> to be set to 1.
(2) Make sure that the Cs origin established state signal CSPENA <F1048.4> in path 2 is 1, and set the Cs axis coordinate establishment request signal CSFI1<G1274.4> and Cs axis contour control switching signal CON<G1027.7> in path 2 to 1.
(3) The absolute and relative coordinates in path 1 and the machine coordinates in path 2 are established.
(4) Upon completion of count-up, the reference position establishment signal ZRFx<F0120.0 to 7> in path 2 becomes 1. So, reset the Cs axis coordinate establishment request signal CSFI1<G1274.4> in path 2 to 0.
(5) Perform FIN processing for the M code in path 1.

Note

- CMR

Set the command multiplier CMR (parameter No. 1820) for the Cs axis to be subjected to composite control to the same value. If it is set to a different value, coordinate establishment does not end normally.

- Axis synchronous control

Feed axis synchronization control cannot be used for the Cs contour control axis.

- Synchronous control

Setting the Cs axis establishment request signal CSFIx <Gn274.4 to 7> to 1 during synchronization control results in alarm PS5346.
After disabling synchronization control, perform Cs coordinate establishment, and then start synchronization control again.

- Composite control

Disabling or enabling composite control during Cs axis coordinate establishment results in alarm PS5346.

- Superimposed Control

Setting the Cs axis coordinate establishment request signal CSFIx <Gn274.4 to 7> to 1 during superimposed control results in alarm PS5346.
After disabling superimposed control, perform Cs coordinate establishment, and then start superimposed control again.

- Spindle command synchronous control

If the master axis is parked, coordinate establishment is performed according to the slave axis machine position.
Set the Cs axis coordinate establishment request signal CSFIx <Gn274.4 to 7> for the master axis to 1.
The Cs axis coordinate establishment alarm signal CSFOx <Fn274.4 to 7> that is output corresponds to the master axis.

- External machine coordinate system shift

Do not shift external machine coordinate systems during Cs axis coordinate establishment.

Signal

Cs axis coordinate establishment request signals CSFI1 to CSFI4<Gn274.4 to 7>

[Classification] Input signal

[Function] These signals perform the Cs axis coordinate establishment (Cs contour control function).

[Operation] When the signals become 1, the control unit behaves as follows:
- If the signals are 1 in the Cs contour control mode, the absolute and machine coordinates are established according to the Cs axis machine position.
- When the signals become 0, the control unit behaves as follows:
The Cs axis coordinate establishment alarm signal CSFOx<Fn274.4 to 7> becomes 0.

Cs axis coordinate establishment alarm signals CSFO1 to CSFO4<Fn274.4 to 7>

[Classification] Output signal

[Function] These signals indicate that Cs axis coordinate establishment has not been completed normally.

[Output cond.] These signal is 1 in the following case :
- Cs axis coordinate establishment has not been completed normally.

These signal is 0 in the following case :
- The Cs contour control mode is exited, or
- The corresponding Cs axis coordinate establishment request signal CSFIx<Gn274.4 to 7> becomes 0.

Cs axis origin established state signals
First spindle: CSPENA<Fn048.4>, Second spindle: CSPENB<Fn052.4>, Third spindle: CSPENC<Fn171.4>, Fourth spindle: CSPEND<Fn269.4>

[Classification] Output signal

[Function] These signals indicate that Cs coordinate establishment processing is available.

[Output cond.] These signal is 1 in the following case :

- A reference position return is completed normally in the Cs contour control mode.

These signal is 0 in the following case :

- A spindle alarm is detected or the spindle motor rotation speed exceeds the maximum allowable rotation speed (specified in parameter No. 4020).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn274	CSFI4	CSFI3	CSFI2	CSFI1				

	#7	#6	#5	#4	#3	#2	#1	#0
Fn048				CSPENA				
Fn052				CSPENB				
Fn171				CSPENC				
Fn269				CSPEND				
Fn274	CSFO4	CSFO3	CSFO2	CSFO1				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1005								ZRNx

[Input type] Parameter input

[Data type] Bit axis

#0 ZRNx If a move command other than G28 is specified by automatic operation when no reference position return is performed yet after the power is turned on:

0: The alarm PS0224, "ZERO RETURN NOT FINISHED." is issued.

1: Operation is performed without issuing an alarm.

NOTE

1 The state in which a reference position has not been established refers to the following state:
- When an absolute position detector is not used and reference position return has not been performed even once after power-up
- When an absolute position detector is used and the association of the machine position with the position detected with the absolute position detector has not been completed (See the description of bit 4 (APZx) of parameter No. 1815.)

2 When the Cs axis coordinates are to be set up, set this parameter to 0.

3 To use a function that establishes the reference point and makes a movement with a command other than G28, such as an axis of Cs contour control, set this parameter to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
3712						CSF		

[Input type] Parameter input
[Data type] Bit

#2 CSF In the Cs contour control mode, the function for setting machine coordinates and absolute coordinates based on the machine position of the spindle if the origin is already set up is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
4353			CSPTRE					

[Input type] Parameter input
[Data type] Bit spindle

#5 CSPTRE Specifies whether to enable Cs axis positional data transfer, as follows:
0: Disable.
1: Enable.

To use this function, reset bit 7 (RFCHK3) of parameter No. 4016 to 0.

Alarm and message

Number	Message	Description
PS5346	REFERENCE POSITION RETURN IS NOT PERFORMED	The coordinate establishment of the Cs contour control axis is not made. Perform a manual reference position return. 1. When Cs coordinate establishment is made for the Cs-axis for which the Cs-axis reference position status signal CSPENx is 0 2. When positional information is not sent from the spindle amplifier 3. When the servo off state is entered during the start of Cs-axis coordinate establishment 4. When the Cs-axis is subjected to synchronous control or superposition control 5. When the emergency stop state is entered during coordinate establishment 6. When an attempt is made to release composite control for the Cs axis being subjected to coordinate establishment 7. When an attempt is made to start synchronous, composite, or superposition control for the Cs axis being subjected to coordinate establishment.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	Cs contour control

10.11.5 Cs Contour Control Manual High-Speed Reference Position Return

Overview

This function makes it possible to perform high-speed type reference position return as manual reference position return after the reference position for the Cs contour control axis is established (ZRF <Fn120> = 1).

Explanation

As manual reference position return after the reference position for the Cs contour control axis is established,

- When the bit 6 (CHM) of parameter No. 3729 is set to 0:
  Spindle orientation is performed.
- When the bit 6 (CHM) of parameter No. 3729 is set to 1:
  High-speed type reference position return is performed.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3729		CHMs						

[Input type] Parameter input
[Data type] Bit spindle

#6 CHMs Manual reference position return after the reference position for the Cs contour control axis is established is performed as:
0: Spindle orientation operation.
1: High-speed type of reference position return operation.

Limitation

- Manual reference position return before the reference position is established

If manual reference position return is performed on the Cs contour control axis before the reference position is established, the return is performed as spindle orientation even if bit 6 (CHM) of parameter No. 3729 is set to 1.

10.12 MULTI-SPINDLE CONTROL

Overview

In addition to the first spindle, the second to fourth spindles can be controlled using an S command from the CNC.
Spindle commands are specified using a single S command as conventionally done. A spindle is selected depending on the signal from the PMC or the address P command.
Gear change between two stages can be made for the second, third, and fourth spindles in the same manner as for the first spindle.
A maximum speed can be set for each spindle, and the speed of each spindle can be clamped to the corresponding maximum speed (according to the setting in parameter No. 3772).
The position coder interfaces for the second to fourth spindles can be selected and added.
The first to fourth position coders are selected by signals from the PMC.

> **NOTE**
> Multi-spindle control can be used by setting bit 3(MSP) of parameter No.8133 to 1 and bit 1(EMS) of parameter No.3702 to 0.

Explanation

- Multi-spindle control in machining center

M

- Only when the T-type gear change is selected (when the constant surface speed control function is enabled(bit 0(SCC) of parameter No.8133 is 1)or when bit 4 (GTT) of parameter No. 3706 is set to 1), multi-spindle control is enabled.
- For information about rigid tapping, see "RIGID TAPPING".

- Control methods

There are three types of multi-spindle control methods: a method that can use the SIND (spindle motor speed control by the PMC) function only for the first spindle (TYPE-A), a method that can use the SIND function for the four spindles independently (TYPE-B), and a method that is equivalent to TYPE-B and can use address P to select the spindle to be controlled by the S command.

Basic control 1 (Common to TYPE-A and TYPE-B)

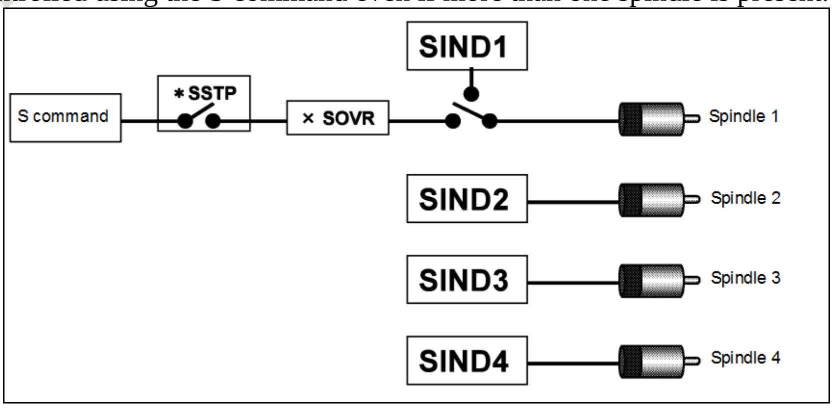
An S command is sent as a speed command to each spindle selected, using a spindle selection signal (SWS1 to SWS4 <Gn027.0-2>, <Gn026.3>). Each spindle rotates at the specified speed. If a spindle is not sent a spindle selection signal, it continues to rotate at its previous speed. This allows the spindles to rotate at different speeds at the same time.

Basic control 2 (Common to TYPE-A, TYPE-B, and the method of making spindle selection by address P)

Each spindle also has a spindle stop signal (*SSTP1 to *SSTP4 <Gn027.3-5>, <Gn026.6>) to stop its rotation; an unused spindle can still be stopped.
There is a spindle enable signal to control each spindle; ENB <Fn001.4> controls the first spindle, while ENB2 to ENB4 <Fn038.2, 3>, <Fn039.1> control the second and third spindles, respectively.
The selection of the position coder by the position coder select signals PC2SLC, PC3SLC, and PC4SLC <Gn028.7>, <Gn026.0>, <Gn026.1> is used to select between the first to fourth position coders.

When multi-spindle control is disabled

When bit 1 (EMS) of parameter No. 3702 is 1, multi-spindle control is disabled. In this case, only the first spindle can be controlled using the S command even if more than one spindle is present.



Multi-spindle control (TYPE-A)

When bit 2 (MSI) of parameter No. 3709=0, TYPE-A is used.
When the first spindle is selected with the SWS1 signal, the SIND signal <Gn033.7> is used to determine whether the spindle analog voltage is controlled by the PMC or CNC; then signals R01I to R12I

<Gn032.0 to Gn033.3> are used to set that spindle's analog voltage. These signals do not affect the second to fourth spindles.
The PMC-based polarity (rotation direction) control signals SGN<Gn033.5> and SSIN <Gn033.6> will function for any spindle selected by SWS1 to SWS4.
The concept of Type A multi-spindle control is outlined below (Fig. 10.12 (a)).

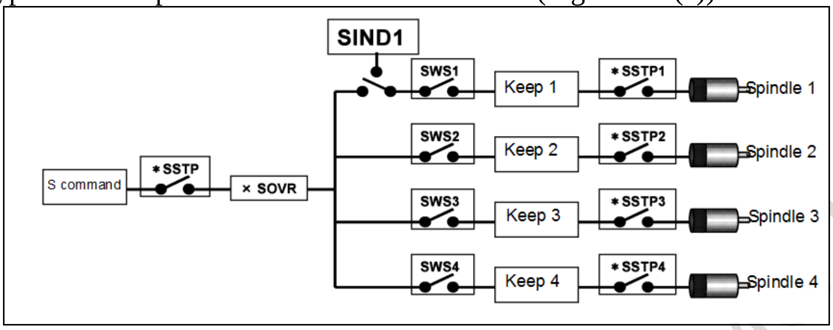


Fig. 10.12 (a)

Multi-spindle control (TYPE-B)

Select Type B control by setting bit 2 (MSI) of parameter No. 3709 to 1.
Each spindle has its own SIND, SSIN and SGN signals.
Each of these signals functions regardless of selection state of the spindle selection signals (SWS1 to SWS4).
When either the spindle selection signals (SWS1 to SWS4) or the SIND signal for the first, second, third, or fourth spindle is set to 1, the polarity control signals SSIN, SGN will function.
The concept of Type B multi-spindle control outlined below (Fig. 10.12 (b)).

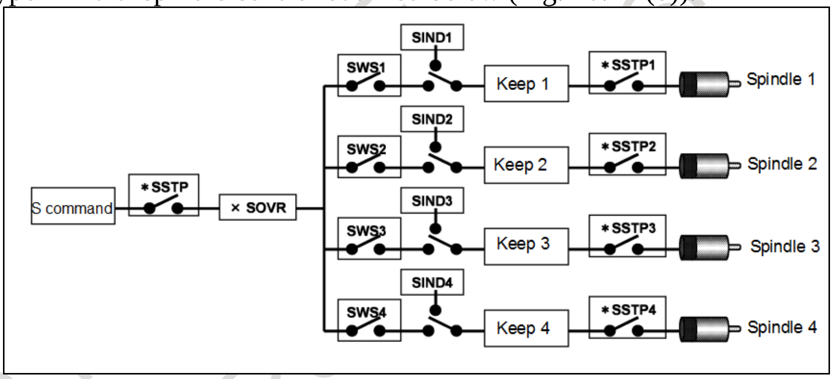


Fig. 10.12 (b)

Multi-spindle control: when a spindle is selected by address P

If bit 3 (MPP) of parameter No. 3703 is set to 1, a spindle is selected by address P. This control method is basically the same as TYPE-B. The first, second, third, and fourth spindles each have their own SIND, SSIN, and SGN signals. A spindle is selected by the P command instead of the spindle selection signals (SWS1 to SWS4). The relationship between the P code and the selected spindle is set in parameter No. 3781.
Polarity (rotation direction) control signals SSIN and SGN for each spindle are valid only for the spindle selected by the P command or for the spindle of which SIND signal is 1.
The concept of multi-spindle control (spindle selection by address P) is outlined below (Fig. 10.12 (c)).

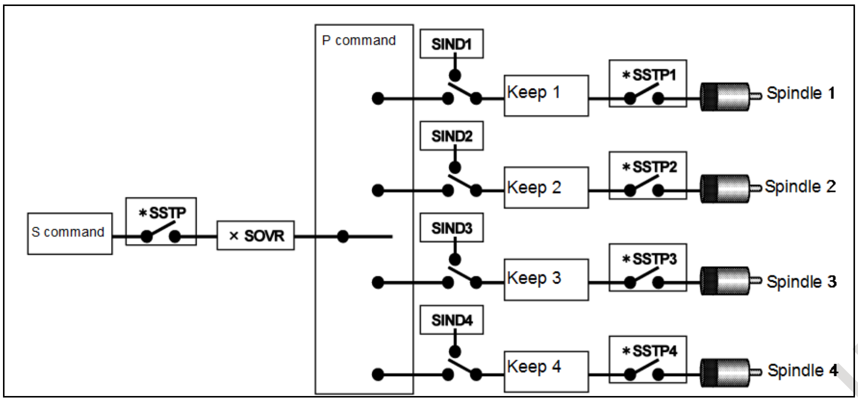


Fig. 10.12 (c)

In case of multi-path control, the P code becomes valid in all paths. For example, suppose that the P code to select the first spindle of path 2 is set to 21. When the following is specified in path 1:

S1000 P21;

Then, 1000 is specified for the first spindle in path 2. Therefore, the same P code cannot be used even in different paths.

- A spindle is selected by address P

Restrictions

The command for spindle selection by address P and the G code or M code including address P cannot be commanded on the same block. If you use the same address, it may result in the machine behaving unexpectedly. Command on separate block. About the function including address P, refer to the appendix “LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE PROGRAM COMMAND”.

- Extended spindle name

Usually, S is used as commands for spindles. If the conditions listed below are all satisfied, extended spindle names can be used. Extended spindle names can consist of up to three characters beginning with S as the first spindle name character, and use of an extended spindle name allows a command to be executed for a specific spindle.

- The serial (analog) spindle function is enabled.
- The multi-spindle control is enabled.
- Bit 0 (EEA) of parameter No. 1000 is 1.
- Bit 3 (MPP) of parameter No. 3703 is 1.
- Bit 1 (ESN) of parameter No. 3798 is 1.
- Bit 4 (GTT) of parameter No. 3706 is 1. (for machining center system only)

As the second and third characters of spindle names, any characters from 0 to 9 and from A to Z may be set in ASCII code. In spindles, the third spindle name character is not valid unless the second spindle name character is set. If a character 0 to 9 is set as the second spindle name character, a character A to Z must not be set as the third spindle name character.

If a spindle name ends with a numeric character, ‘=‘ is required between the spindle name and a specified value.

	Spindle name 1st character (fixed)	Spindle name 2nd character (No.3738)	Spindle name 3rd character (No.3739)
Setting value	‘S’	‘0’ to ‘9’	‘0’ to ‘9’
		‘A’ to ‘Z’	‘0’ to ‘9’ ‘A’ to ‘Z’
Correct example <1>	‘S’	1	1
Correct example <2>		A	1
Correct example <3>		A	B
Incorrect example		1	A

In case of multi-path control, extended spindle names become valid in all paths. For example, suppose that the first spindle of path 2 is named SA. When the following is specified in path 1:

SA1000;

Then, 1000 is specified for the first spindle in path 2. Therefore, the same extended spindle name cannot be used even in different paths.

Restrictions

More than one extended spindle name cannot be specified within the same block. Also, ‘S’ and an extended spindle name cannot be specified together within the same block. If they are specified together, P/S alarm No. 333 is issued.
When extended spindle names are used, subscripts (MAIN (in parameter No. 3718), SUB (in parameter No. 3719) cannot be used.
When the custom macro function is enabled, the same extended spindle names as reserved words cannot be used. They are regarded as reserved words.
The target spindle for rigid tapping cannot be selected by specifying an extended spindle name.

Multi-spindle address P signal

When extended spindle names are used, the value in parameter No. 3781 corresponding to the spindle number of the spindle name specified last is output to MSP00 to MSP15 <Fn160.0 to Fn161.7>. If a command for a spindle has not been specified even once, the value set in parameter No. 3775 is output.

Supplement

When extended spindle names are used, the value of the spindle command specified last is indicated as the modal indication of S.
When the serial (analog) spindle function and multi-spindle control are enabled, the method for selecting the target spindle is determined according to the settings of bit 3(MPP) of parameter No. 3703 and bit 1 (ESN) of parameter No. 3798 as follows:

Bit 3 (MPP) of parameter No. 3703	Bit 1 (ESN) of parameter No. 3798	Method for selecting target
0	0	Signal selection
0	1	Signal selection
1	0	P command (S_ P_)
1	1	Specification of extended spindle name or P command (S_ P_)

If MPP and ESN are both set to 1, both the S_P_ command and extended spindle names can be used.

NOTE

Address S indicates a command for all spindles. When multiple spindles are used, and bit 3 (MPP) of parameter No. 3703 is 1, a particular spindle must be specified by using address P and so on. If spindle name S is set when extended spindle names are used, S indicates a particular spindle (except the command for clamping at the maximum spindle speed in constant surface control (G50(G92)S_)).

This means that if spindle name S is set when extended spindle names are used, a command specifying S only is a command for the spindle named S. (Even when P is specified in the same block, it is ignored.) When S is not set in a spindle name, a command specifying S only is regarded as a command for the spindle for which a command has been issued last. (This is the same as the operation of a command specifying only S when multiple spindles are used, extended spindle names are not used, and bit 2 (MPA) of parameter No. 3706 is set to 1.)

Example: When extended spindle names are used, and commands with extended spindle names and S_P_ commands are used together

1. When the first spindle is SA, the second spindle is SB, and 1 is set for the first spindle and 2 is set for the second spindle in parameter No. 3781

<u>If bit 2 (MPA) of parameter No. 3706 is 0</u>

N10 M03;
N20 SA100; → 100 is specified for 1st spindle.
N30 SB200; → 200 is specified for 2nd spindle.
N40 S200P1; → 200 is specified for 1st spindle.
N50 S300; → P/S alarm No. 5305 is issued.
N60 S200P2;
N70 S300;
N80 SA400;
N90 S500;
N100 SB400;
N110 S500;

<u>If bit 2 (MPA) of parameter No. 3706 is 1</u>

N10 M03;
N20 SA100; → 100 is specified for 1st spindle.
N30 SB200; → 200 is specified for 2nd spindle.
N40 S200P1; → 200 is specified for 1st spindle.
N50 S300; → 300 is specified for 1st spindle.
N60 S200P2; → 200 is specified for 2nd spindle.
N70 S300; → 300 is specified for 2nd spindle.
N80 SA400; → 400 is specified for 1st spindle.
N90 S500; → 500 is specified for 1st spindle.
N100 SB400; → 400 is specified for 2nd spindle.
N110 S500; → 500 is specified for 2nd spindle.

NOTE

2. When the first spindle is SA, the second spindle is S, and 1 is set for the first spindle and 2 is set for the second spindle in parameter No. 3781

   If bit 2 (MPA) of parameter No. 3706 is 0

   N10 M03;
   N20 SA100; → 100 is specified for 1st spindle.
   N30 S200P1; → 200 is specified for 2nd spindle.
   N40 S300; → 300 is specified for 2nd spindle.
   N50 S200P2; → 200 is specified for 2nd spindle.
   N60 S300; → 300 is specified for 2nd spindle.
   N70 SA400; → 400 is specified for 1st spindle.
   N80 S500; → 500 is specified for 2nd spindle.

   If bit 2 (MPA) of parameter No. 3706 is 1

   N10 M03;
   N20 SA100; → 100 is specified for 1st spindle.
   N30 S200P1; → 200 is specified for 2nd spindle.
   N40 S300; → 300 is specified for 2nd spindle.
   N50 S200P2; → 200 is specified for 2nd spindle.
   N60 S300; → 300 is specified for 2nd spindle.
   N70 SA400; → 400 is specified for 1st spindle.
   N80 S500; → 500 is specified for 2nd spindle.

Example

- Example of setting spindle names when extended spindle names are used
  No.3738=0,No.3739=0 → 'S'
  No.3738=65,No.3739=0 → 'SA'
  No.3738=66,No.3739=65 → 'SBA'
  No.3738=49,No.3739=0 → 'S1'
  No.3738=49,No.3739=48 → 'S10'
  No.3738=0,No.3739=65 → 'S'

- Specifying spindle names that end with a numeric character
  S1=100
  S11=200

- Specifying spindle names that end with an alphabetical character
  S100 or S=100
  SB200 or SB=200
  SBB300 or SBB=300

Override for individual spindles

Different override values can be applied to individual axes separately at a time. The override values can be selected from between 0% and 254%. This function is made usable by setting bit 3 (MSC) of parameter No. 3713 to 1 for multi-spindle control.

- Specification

When each spindle speed override function (bit 4 (EOV) of parameter No. 3713 =1) is effective, the specifications is as following.

- Override signals for each spindle are following.
  SOV0 to 7: Override signal for the first spindle
  SOV20 to 27: Override signal for the second spindle
  SOV30 to 37: Override signal for the third spindle
  SOV40 to 47: Override signal for the fourth spindle
- Because override is applied to individual axes separately, it is possible to use different values for each axis simultaneously.

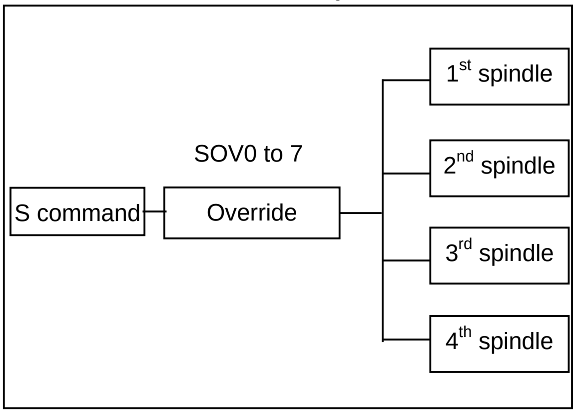


Fig. 10.12 (d) Bit 4 (EOV) of parameter No. 3713 = 0

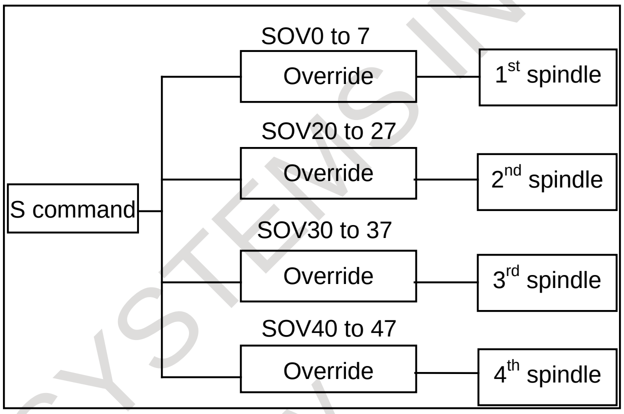


Fig. 10.12 (e) Bit 4 (EOV) of parameter No. 3713 = 1

Also for spindle control in a multi-path system, this function can be used to apply override to individual axes separately. Fig. 10.12 (f), “Bit 4 (EOV) of parameter No. 3713 = 0 (2-path system)” and Fig. 10.12 (g), “Bit 4 (EOV) of parameter No. 3713 = 1 (2-path system)” are examples of using this function for a 2-path system. In these examples, both path 1 and path 2 spindles are controlled by the spindle commands in path 2.

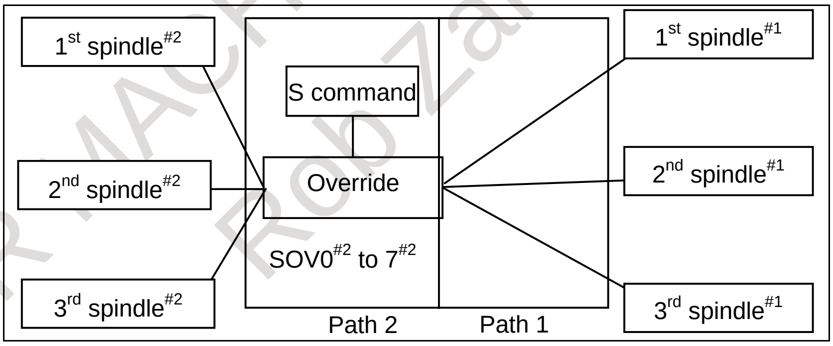


Fig. 10.12 (f) Bit 4 (EOV) of parameter No. 3713 = 0 (2-path system)

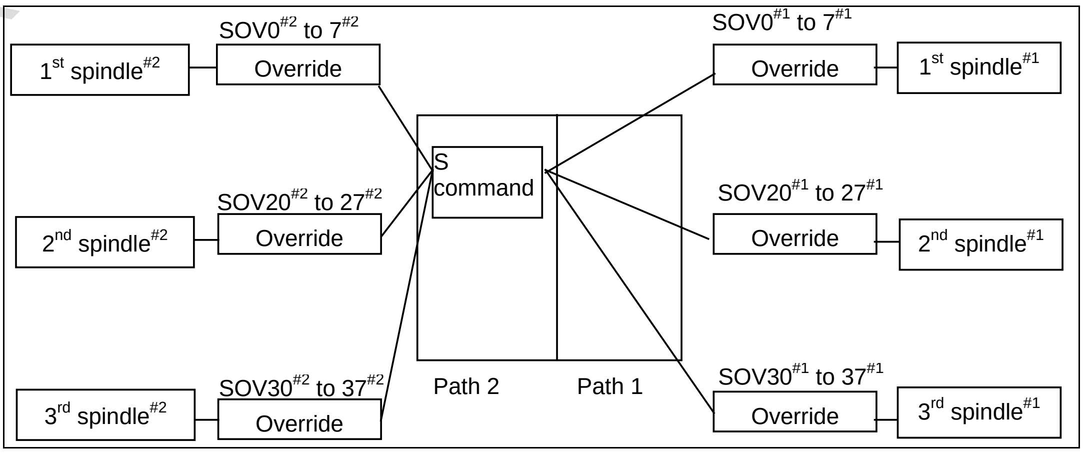


Fig. 10.12 (g) Bit 4 (EOV) of parameter No. 3713 = 1 (2-path system)

- Note

- Setting bit 5 (SOV) of parameter No. 3106 (spindle override value display) causes the override value for the first spindle to be displayed.
- When constant surface speed control is used, override is applied after the spindle rotation speed is clamped to the clamp speed specified by the spindle rotation speed clamp command (for the T series, G50S_, or for the M series, G92S_). Note, therefore, that if an override of 100% or higher is applied, the spindle may rotate faster than the specified clamp speed.
  Bit 5 (SOC) of parameter No. 3708 (for specifying to clamp the spindle rotation speed for constant surface speed control after spindle speed override is put into effect) is invalid. The note given above still applies even if this parameter is set. The spindle rotation speed can be clamped to the clamp speed specified as the spindle rotation speed upper limit (parameter No. 3772).
- When this function is in use, a spindle override of 100% is applied during tapping cycle mode operation (for the M series, G84 and G74 or, for the T series, G84 and G88) or threading operation (for the M series, G33 or, for the T series, G32, G92, and G76). Setting bit 6 (TSO) of parameter No. 3708 to 1 enables spindle override.
- For spindles not subjected to override, keep their spindle override signals set to 100%.
- No spindle override is applied to spindles which are under spindle rotation speed control directed by the PMC.

Position coder selection by address P (bit 6 (MPC) of parameter No. 3713=1)

When the spindle is selected by address P in multi-spindle control (bit 3 (MPP) of parameter No. 3703=1), position coder feedback is automatically selected by address P for feed per revolution, thread cutting, and so on.

- Explanation

When the spindle is selected by address P in multi-spindle control (bit 3 (MPP) of parameter No. 3703=1), the program of spindle selection by address P is as follows.

S_ P_ ;

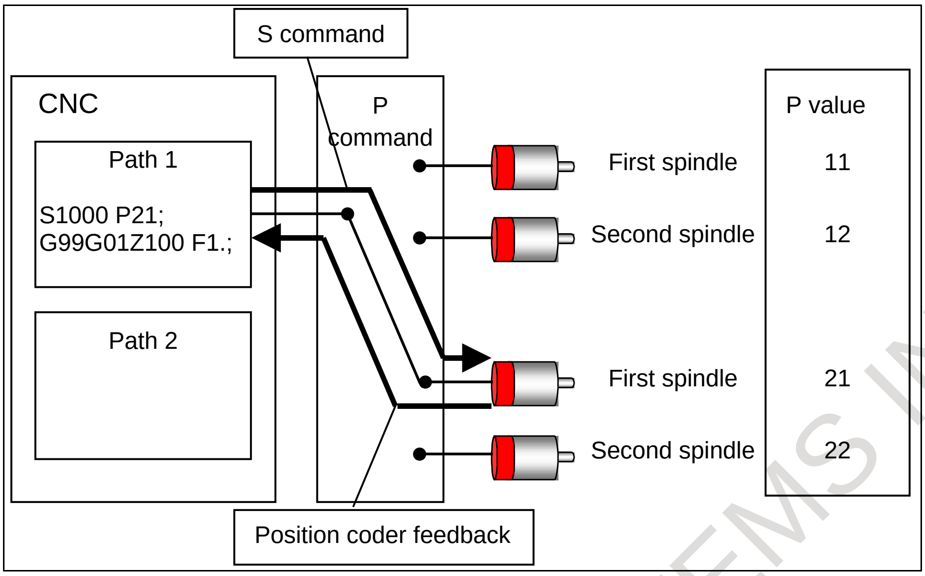
When the bit 6 (MPC) of parameter No. 3713 is set to 1, this command causes the position coder to be automatically selected according to the selected spindle. This condition is the same as the selection of the position coder by the position coder select signals PC2SLC<Gn028.7>, PC3SLC<Gn026.0> and PC4SLC<Gn026.1> and inter-path spindle feed-back signals SLPCA<Gn064.2>, SLPCB<Gn064.3>, SLPCC<Gn403.4>, and SLPCD<Gn403.5>. (These signals are ignored even if they are set from the PMC ladder.)

- Example

In the following program:

```
S1000 P21 ;
G99 G01 Z100. F1. ;
```

(1) The first spindle in path 2 is specified by S1000.
(2) Position coder feed-back for the first spindle in path 2 is selected automatically.
    (Used in feed per revolution)



- Relationship with other functions

Constant surface speed control

The control function for keeping the surface speed constant can be used with any of the four spindles, if the spindle speed is within the range allowable for this function. (If position coders are required, this function is enabled for serial spindles.) The spindle selection signal (SWS1 to SWS4) for the spindle must stay set at 1 during machining using this function.

Spindle speed fluctuation detection

When this function is used with multi-spindle control, multiple position coders can be used. So, when a spindle speed fluctuation is detected, attention must be paid to the states of the position coder selection signals (PC2SLC, PC3SLC, and PC4LC) and spindle selection signals (SWS1 to SWS4)

Actual spindle speed output

The actual spindle speed output function conveys speed information obtained from the position coder specified by the position coder selection signals (PC2SLC, PC3SLC, and PC4SLC) to the PMC.

Polygon turning

Polygon turning rotates a tool axis in phase with the spindle. To perform polygon turning when multi-spindle control issued, select the spindle and the position coder associated with the spindle.

Spindle synchronization, polygon turning between spindles, simplified synchronization control

During spindle synchronization, polygon turning between spindles or simplified synchronization control, the slave spindle operates in phase with the master spindle. Multi-spindle control for the master spindle can be used during synchronization control, but multi-spindle control for the slave spindle is disabled.

Rigid tapping

One of the first to fourth spindles can be selected as the rigid tapping axis according to the states of the spindle selection signals (SWS1 to SWS4) to perform rigid tapping. However, the following restrictions are imposed:

- Set the SWS1 to SWS4 signals before directing rigid tapping;
- Do not switch the SWS1 to SWS4 signals during rigid tapping; and
- Use the appropriate ENB signal (either ENB or ENB2) for the selected spindle as the ENB signal for the rigid tapping PMC sequence.

The spindles not used for rigid tapping can be rotated at a speed specified before rigid tapping starts, or can be stopped.

Ladder sequence (signal operation)

Outline of the CNC processing and the relation with the ladder

On multi-spindle control TYPE-A and TYPE-B, CNC considers S command (program command), signals related to S command (ex. Spindle override) and G-code (ex. G96/G97), and converts to the actual spindle rotation speed. Then CNC sends this value only to the spindle that is selected by spindle selection signals SWS1 to SWS4 <Gn027.0 to Gn027.2 and Gn026.3>. On this document, a series of this processing is called "S renewal processing". S renewal processing is asynchronously executed with the changing of the select/non-select condition of the spindle by ladder sequence. Therefore, the order of the changing of the select/non-select condition of each spindle and S command must be considered by the ladder sequence or the program command according to the necessity.

S command

Ladder can confirm the S command by SF signal. In case of waiting the completion of S renewal processing after the S command, please take time by using the timer of ladder and so on.
A necessary processing time to complete S renewal processing from the S command is 16msec. As for the output condition of SF signal, please refer to parameter No.3705. On T series system, please set EVS (Parameter No.3705#4) to "1" to output SF signal.

Time chart

The example of the time chart when executing the following program is shown.

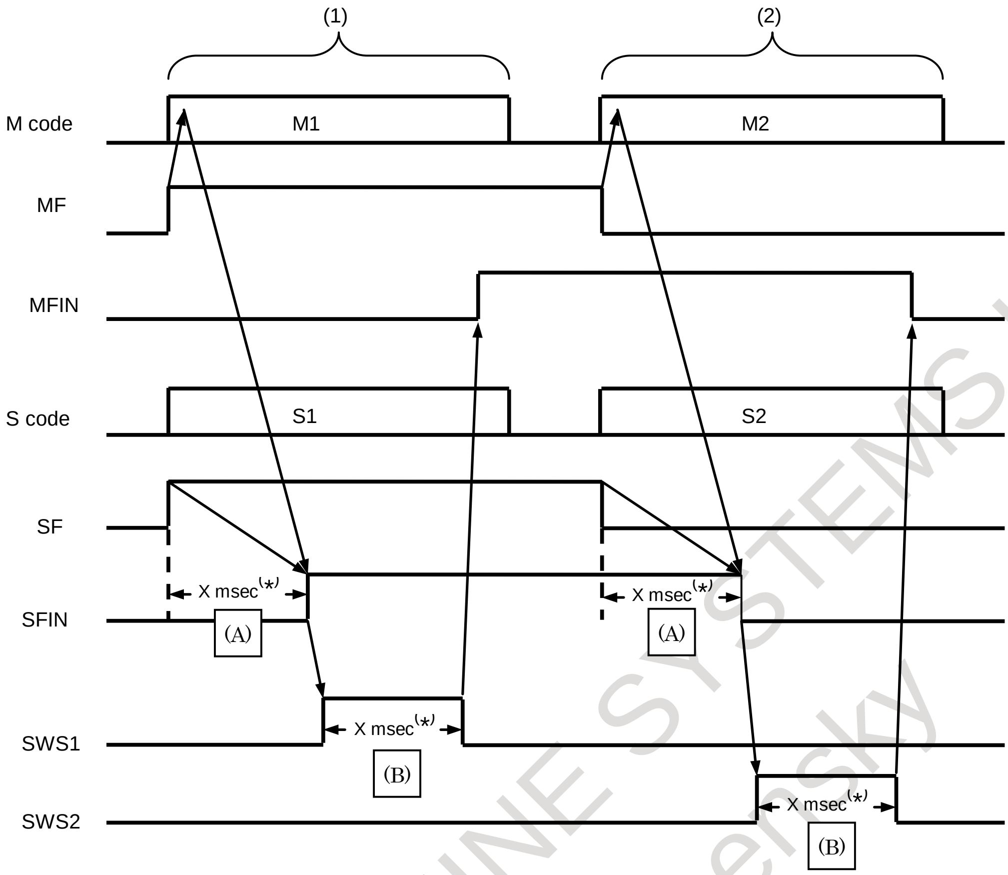
M1_ S1_ ; ………… (1)

M2_ S2_ ; ………… (2)

(1): The program to execute S command of S1_ to the 1st spindle.

(2): The program to execute S command of S2_ to the 2nd spindle. And at this time, the 1st spindle must keep S1_, which was commanded in (1).

The following time chart shows example in case of high-speed M/S/T/B. (bit 7 (MHI) of parameter No.3001)



(*) As for the value of "X msec", refer to the following Appendix 1.

In the time chart, it takes time to start S renewal processing of CNC in (A).
At this time, both SWS1 and SWS2 signals are "0". Therefore, S command is not forwarded to any spindle even when S renewal processing is completed.

It takes time to execute S renewal processing with selecting the objective spindle (SWSx=1) in (B).
At this time, the 1st spindle (or the 2nd spindle) is selected, the renewal S command is forwarded to the objective spindle after completing S renewal processing.

If (B) ends, please return all spindles to non-selected condition (SWSx=0).

(Supplement 1) The (A) part and the (B) part are timer of ladder sequence. As for X msec, please apply the value which suits the condition “X msec > 16 msec ”.

(Supplement 2) While machining by the constant surface control, the spindle must be always selected condition. Please refer to "Relationship with other functions" previously described.

Signal

Spindle selection signals SWS1,SWS2,SWS3,SWS4<Gn027.0,.1,.2>,<Gn026.3>

[Classification] Input signal

[Function] Controls whether S command specified to the NC is output to the spindle or not in multi-spindle control.

SWS1 1 : Outputs a speed command to the first spindle.
0 : Outputs no speed command to the first spindle.

SWS2 1 : Outputs a speed command to the second spindle.
0 : Outputs no speed command to the second spindle.

SWS3 1 : Outputs a speed command to the third spindle.
0 : Outputs no speed command to the third spindle.

SWS4 1 : Outputs a speed command to the fourth spindle.
0 : Outputs no speed command to the fourth spindle.

Individual spindle stop signals *SSTP1,*SSTP2,*SSTP3,*SSTP4<Gn027.3,.4,.5>,<Gn026.6>

[Classification] Input signal

[Function] Effective only to multi-spindle control, each spindle can be stopped by this signal.

*SSTP1 1 : Does not set 0 min^{-1} for output to the first spindle.
0 : Sets 0 min^{-1} for output to the first spindle.

*SSTP2 1 : Does not set 0 min^{-1} for output to the second spindle.
0 : Sets 0 min^{-1} for output to the second spindle.

*SSTP3 1 : Does not set 0 min^{-1} for output to the third spindle.
0 : Sets 0 min^{-1} for output to the third spindle.

*SSTP4 1 : Does not set 0 min^{-1} for output to the fourth spindle.
0 : Sets 0 min^{-1} for output to the fourth spindle.

Gear select signals GR11<Gn028.1>,GR12<Gn028.2>,GR21<Gn029.0>,GR22<Gn029.1>, GR31<Gn029.2>,GR32<Gn029.3>,GR41<Gn031.4>,GR42<Gn031.5>

[Classification] Input signal

[Function] Gear selection signals when multi-spindle control is equipped. These signals post the gear stage currently selected (4 stages).

Gear	GRs1	GRs2
Gear 1	0	0
Gear 2	1	0
Gear 3	0	1
Gear 4	1	1

(s is 1, 2, 3, or 4.)

Position coder selection signals PC2SLC<Gn028.7>,PC3SLC<Gn026.0>,PC4SLC<Gn026.1>

[Classification] Input signal

[Function] These signals select the position coder of the serial spindle used for multi-spindle control.

Position coder to be selected	PC2SLC	PC3SLC	PC4SLC
First position coder	0	0	0
Second position coder	1	0	0
Third position coder	0	1	0
Fourth position coder	0	0	1

If PC2SLC, PC3SLC, and PC4SLC are set to 1, the position coder of a lower number is selected.

When the second, third, or fourth position coder is not installed, do not switch this signal.

Always set 0 for first position coder.

Spindle enable signals
ENB<Fn001.4>,ENB2<Fn038.2>,ENB3<Fn038.3>,ENB4<Fn039.1>

[Classification] Output signal

[Function] These signals inform PMC of whether or not to perform output to the second to fourth spindles in multi-spindle control. The signals are used as a condition to stop the analog spindle, and are also used for a PMC ladder sequence that is associated with rigid tapping. (See "RIGID TAPPING.")

[Output cond.]

ENB	Enables output of command other than 0 to the first spindle control unit. Outputs 0 to the first spindle control unit.
ENB2	Enables output of command other than 0 to the second spindle control unit. Outputs 0 to the second spindle control unit.
ENB3	Enables output of command other than 0 to the third spindle control unit. Outputs 0 to the third spindle control unit.
ENB4	Enables output of command other than 0 to the fourth spindle control unit. Outputs 0 to the fourth spindle control unit.

Spindles control signal by PMC
For 1st spindle SIND,SSIN,SGN<Gn033.7,.6,.5>,R01I-R12I<Gn032.0-Gn033.3>
For 2nd spindle SIND2,SSIN2,SGN2<Gn035.7,.6,.5>,R01I2-R12I2<Gn034.0-Gn035.3>
For 3rd spindle SIND3,SSIN3,SGN3<Gn037.7,.6,.5>,R01I3-R12I3<Gn036.0-Gn037.3>
For 4th spindle SIND4,SSIN4,SGN4<Gn273.7,.6,.5>,R01I4-R12I4<Gn272.0-Gn273.3>

[Classification] Input signal

[Function] The spindle motor of each spindle can be controlled by issuing commands from the PMC. The speed command and polarity (rotation direction) of the spindle motor can be controlled. Usually, CNC commands are used to specify a speed and polarity. By using these signals, whether commands issued from the CNC or PMC are to be used for this control can be selected.

Even when multi-spindle control is not being used, the signals can be used to control the second to fourth spindles.

When multi-spindle control is being used, and TYPE-A is selected (bit 2 (MSI) of parameter No. 3709 is set to 0), the signals for the second to fourth spindles cannot be used.

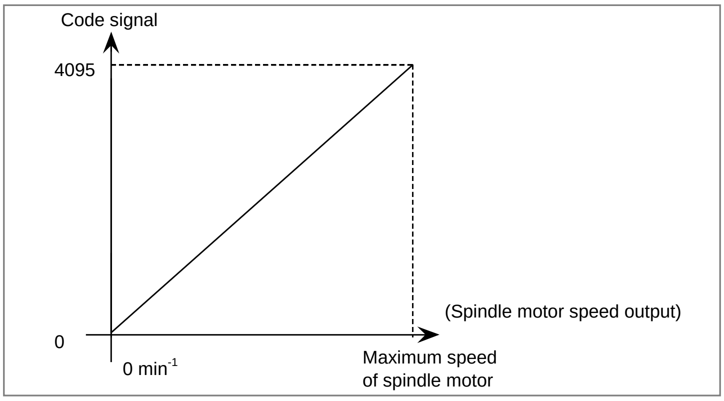
For details of each signal, see "SPINDLE OUTPUT CONTROL BY THE PMC."

S12-bit code signals
R01O to R12O<Fn036.0 to Fn037.3>,
R01O2 to R12O2<Fn200.0 to Fn201.3>,
R01O3 to R12O3<Fn204.0 to Fn205.3>,
R01O4 to R12O4<Fn270.0 to Fn271.3>,

[Classification] Output signal

[Function] If bit 3 (MRS) of parameter No. 3709 is 1, the spindle speed command value calculated by the CNC is converted to a code signal from 0 to 4095, and the conversion result is output to a spindle. If MRS is set to 0, the data of the currently selected spindle (when more than one spindle is selected, the spindle with the smallest number) is output only to R01O to R12O for the first spindle.

[Output cond.] The relationship between the spindle speed command output value (the value calculated by the CNC) and the value output to these signals is plotted in the graph shown below. Note that for either a serial or analog spindle, the output value on these signals is a value 0 to 4095 as a result of conversion from the spindle speed command output value calculated by the spindle control function of the CNC; the output value on these signals is not the actual output value.



Address P signals MSP00 to MSP15<Fn160.0 to Fn161.7>

[Classification] Output signal

[Function] The P value specified last by the S_P_; command is output.
When extended spindle names are used, the value in parameter No. 3781 corresponding to the spindle number of the spindle name specified last is output.

[Output cond.] When multi-spindle control by using address P is enabled (bit 3 (MPP) of parameter No. 3703 is 1), the P value specified in the S_P_; command is output. When no S_P_; command has been specified even once since power-up, the initial P value set in parameter No. 3775 is output.
When extended spindle names are used, the value in parameter No. 3781 corresponding to the spindle number of the spindle name specified last is output. When no spindle name has been specified even once, the value set in parameter No. 3775 is output.

1st spindle speed override signal SOV0 to SOV7<Gn030>
2nd spindle speed override signal SOV20 to SOV27<Gn376>
3rd spindle speed override signal SOV30 to SOV37<Gn377>
4th spindle speed override signal SOV40 to SOV47<Gn378>

[Classification] Input signal

[Function] If bit 4 (EOV) of parameter No. 3713 = 1, an override of 0% to 254% of the rotation speed specified by the CNC can be applied to individual spindles separately.

[Operation] Specify an override value, using an 8-bit binary number.
If all bits are 1, however, an override of 0% is assumed.

> **NOTE**
> If bit 4 (EOV) of parameter No. 3713 = 0, the override signals SOV0 to SOV7 are applied to all spindles that receive speed commands.
> The signals SOV20 to SOV27, SOV30 to SOV37, and SOV40 to SOV47 are invalid.

Spindle command path specification signal SPSP<Gn536.7>

[Classification] Input signal

[Function] Turns spindle command path specification ON or OFF.

[Operation] If a command based on address P (bit 3 (MPP) of parameter No. 3703 = 1 and bit 1 (ESN) of parameter No. 3798 = 0) or a command based on an extended spindle name (bit 3 (MPP) of parameter No. 3703 = 1, bit 0 (EEA) of parameter No. 1000 = 1, and bit 1 (ESN) of parameter No. 3798 = 1) is used in multi-spindle control:

- Setting this signal to 1 enables path specification for spindle commands.
  Spindle commands can be issued from the path specified by parameter No. 11090.
  If a spindle command is issued from a path other than the path, PS5305 alarm is issued.
- Setting this signal to 0 disables path specification for spindle commands.
  Spindle commands can be issued from individual paths no matter what the setting of parameter No. 11090.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn026		*SSTP4			SWS4		PC4SLC	PC3SLC
Gn027			*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
Gn028	PC2SLC					GR2	GR1	
Gn029		*SSTP			GR32	GR31	GR22	GR21
Gn030	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0
Gn031			GR42	GR41				
Gn032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
Gn033	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
Gn034	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2
Gn035	SIND2	SSIN2	SGN2		R12I2	R11I2	R10I2	R09I2
Gn036	R08I3	R07I3	R06I3	R05I3	R04I3	R03I3	R02I3	R01I3
Gn037	SIND3	SSIN3	SGN3		R12I3	R11I3	R10I3	R09I3
Gn272	R08I4	R07I4	R06I4	R05I4	R04I4	R03I4	R02I4	R01I4
Gn273	SIND4	SSIN4	SGN4		R12I4	R11I4	R10I4	R09I4
Gn376	SOV27	SOV26	SOV25	SOV24	SOV23	SOV22	SOV21	SOV20
Gn377	SOV37	SOV36	SOV35	SOV34	SOV33	SOV32	SOV31	SOV30
Gn378	SOV47	SOV46	SOV45	SOV44	SOV43	SOV42	SOV41	SOV40
Gn536	SPSP							

	#7	#6	#5	#4	#3	#2	#1	#0
Fn001				ENB				
Fn036	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
Fn037					R12O	R11O	R10O	R09O
Fn038					ENB3	ENB2		
Fn039							ENB4	
Fn160	MSP07	MSP06	MSP05	MSP04	MSP03	MSP02	MSP01	MSP00
Fn161	MSP15	MSP14	MSP13	MSP12	MSP11	MSP10	MSP09	MSP08
Fn200	R08O2	R07O2	R06O2	R05O2	R04O2	R03O2	R02O2	R01O2

Fn201					R12O2	R11O2	R10O2	R09O2
Fn204	R08O3	R07O3	R06O3	R05O3	R04O3	R03O3	R02O3	R01O3
Fn205					R12O3	R11O3	R10O3	R09O3
Fn270	R08O4	R07O4	R06O4	R05O4	R04O4	R03O4	R02O4	R01O4
Fn271					R12O4	R11O4	R10O4	R09O4

Parameter

Parameters related to the first spindle and first position coder are the same as those used conventionally. The following explains the parameters added in connection with this function and related parameters:

	#7	#6	#5	#4	#3	#2	#1	#0
1000								EEA

[Input type] Parameter input
[Data type] Bit

#0 EEA An extended axis name and extended spindle name are:
0: Invalid
1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
3106			SOV					

[Input type] Setting input
[Data type] Bit

#5 SOV A spindle override value is:
0: Not displayed.
1: Displayed.

NOTE
This parameter is valid only when bit 2 (DPS) of parameter No. 3105 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
3702							EMS	

[Input type] Parameter input
[Data type] Bit path

#1 EMS The multi-spindle control is:
0: Used.
1: Not used.

	#7	#6	#5	#4	#3	#2	#1	#0
3703					MPP			

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#3 MPP In multi-spindle control, a spindle selection using a programmed command instead of using the signals (SWS1 to SWS4<Gn027.0 to Gn027.2, Gn026.3>) is:

0: Not made.
1: Made.

> **NOTE**
> When this parameter is set to 1, set parameter No. 3781 at the same time.

	#7	#6	#5	#4	#3	#2	#1	#0
3706						MPA		
				GTT		MPA		

[Input type] Parameter input
[Data type] Bit path

#2 MPA If a spindle is to be selected using a P command (with bit 3 (MPP) of parameter No. 3703 set to 1) in multi-spindle control, and a P command is not specified together with an S command:

0: The alarm PS5305, “ILLEGAL SPINDLE NUMBER” is issued.
1: The last P specified by S_ P_; (by S_ P_; specified for the path in case of a multi-path system) is used. If P is not specified even once after power-up, the value of parameter No. 3775 is used.

> **NOTE**
> This parameter is valid only when bit 3 (MPP) of parameter No. 3703 is set to 1.

#4 GTT Spindle gear selection method is:

0: Type M.
1: Type T.

> **NOTE**
> 1 This parameter is effected for M system.
> 2 M type
> The gear selection signal is not input. The CNC selects a gear based on the speed range of each gear set by a parameter beforehand according to S codes, and the selected gear is posted by outputting the gear selection signal. Moreover, the spindle speed matching the gear selected by the output gear selection signal is output.
> T type
> The gear selection signal is input. The spindle speed matching the gear selected by this signal is output.
> 3 When the constant surface speed control is enabled(bit 0(SSC) of parameter No.8133 is 1), type T is selected, regardless of whether this parameter is specified.

NOTE

4 When type T spindle gear switching is selected, the following parameters have no effect:
No.3705#2(SGB), No.3751, No.3752,
No.3705#1(GST), No.3705#3(SGT), No.3761, No.3762,
No.3705#6(SFA), No.3735, No.3736
On the other hand, parameter No. 3744 becomes usable.

5 When multi spindle control is used, select to T type.

	#7	#6	#5	#4	#3	#2	#1	#0
3708		TSO	SOC					

[Input type] Parameter input
[Data type] Bit path

#5 SOC During constant surface speed control (G96 mode), the speed clamp by the maximum spindle speed clamp command (G92 S_; (G50 for G code system A of lathe system)) is carried out:
0: Before spindle speed override.
1: After spindle speed override.
If this parameter is set to 0, the spindle speed may exceed the maximum spindle speed (numeric value following S in G92 S_; (G50 for G code system A of lathe system)).
If this parameter is set to 1, the spindle speed is limited to the maximum spindle speed.
The spindle speed is limited to the upper limit of spindle speed specified in parameter No. 3772, irrespective of the setting of this parameter.

#6 TSO During a threading or tapping cycle, the spindle override is:
0: Disabled (tied to 100%).
1: Enabled.

NOTE

During rigid tapping, the override is tied to 100%, irrespective of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3709					MRS	MSI		

[Input type] Parameter input
[Data type] Bit path

#2 MSI In multi-spindle control, the SIND signal is valid
0: Only when the first spindle is valid (SIND signal for the 2nd, 3rd spindle becomes ineffective) (TYPE-A)
1: For each spindle irrespective of whether the spindle is selected (Each spindle has its own SIND signal). (TYPE-B)

#3 MRS When the actual spindle speed signals and S 12-bit code signals are output in multi-spindle control:
0: The signals common to the first spindle and second spindle are used, and the signals for the spindle selected by the spindle selection signal are output.
1: The signals for the first spindle and the signals for the second spindle are output separately.

	#7	#6	#5	#4	#3	#2	#1	#0
3713		MPC		EOV	MSC			

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#3 MSC Multi-spindle control TYPE-C is:
0: Not used.
1: Used.

> **NOTE**
> If parameter MSC and bit 2 (MSI) of parameter No. 3709 for multi-spindle control TYPE-B) are set to 1 at the same time, multi-spindle control TYPE-C is enabled.

#4 EOV Each spindle speed override is:
0: Not used.
1: Used.

> **NOTE**
> Multi-spindle control TYPE-C (bit 3 (MSC) of parameter No. 3713=1) is necessary to use this function.

#6 MPC When a spindle is selected with address P in a program during multi-spindle control (bit 3 (MPP) of parameter No. 3703 is set to 1), position coder feedback used for thread cutting, feed per revolution, and so forth is:
0: Not changed automatically according to the selected spindle.
1: Changed automatically according to the selected spindle.

> **NOTE**
> Setting this parameter produces the same effects as when position coder select signals PC2SLC<Gn028.7>, PC3SLC<Gn026.0>, and PC4SLC<Gn026.1>, inter-path spindle feedback signals SLPCA<Gn064.2>, SLPCB<Gn064.3>, SLPCC<Gn403.4>, and SLPCD<Gn403.5> are set.
> At this time, even when an attempt to set these signals is made by a PMC ladder, these signal operations are ignored.

3718	Subscript for display of a serial spindle (main spindle) or analog spindle

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 122

Set a subscript to be added to spindle speed display on a screen such as the position display screen.
Used when the spindle switching function is not used, or used for the main spindle when the spindle switching function is used.

NOTE
This parameter is invalid when an extended spindle name is used.

3719	Subscript for display of a serial spindle (sub-spindle)

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 122

Set a subscript to be added to spindle speed display on a screen such as the position display screen.
Used for the sub-spindle when the spindle switching function is used.

NOTE
This parameter is invalid when an extended spindle name is used.

3720	Number of position coder pulses

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Number of pulses
[Valid data range] Serial spindle
: 0, 4096 When bit 7 (FBP) of parameter No.3716 is set to 1: $4096\times2^{\wedge14}$
Spindle control with servo motor
: 0, 4096 When bit 7 (FBP) of parameter No.3716 is set to 1: $4096\times2^{\wedge14}$
Analog spindle
: 0, 4096 When bit 7 (FBP) of parameter No.3716 is set to 1: from 0 to 400000

Set the number of position coder pulses.
In analog spindle, 4096 is set to parameter automatically if parameter No.3720 is set to 0 or less.
In serial spindle or spindle control with servo motor, the setting method is changed by setting of the bit 7 (FBP) of parameter No.3716.

- When the bit 7 (FBP) of parameter No.3716 is set to 0:
  4096 is set to parameter automatically if parameter No.3720 is set to 0 or less.
- When the bit 7 (FBP) of parameter No.3716 is set to 1:
  $4096\times2^{\wedge14}$ is set to parameter automatically.

3721	Number of gear teeth on the position coder side

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 9999

Set the number of gear teeth on the position coder side in speed control (such as feed per revolution).

3722	Number of gear teeth on the spindle side

[Input type] Parameter input
[Data type] Word spindle

[Valid data range] 0 to 9999

Set the number of gear teeth on the spindle side in speed control (such as feed per revolution).

> ⚠ **CAUTION**
> 1 Parameter No.3721 and No.3722 can not be used for thread cutting.
> If thread cutting was done by setting the any value on these parameters, the correct screw can't be cut.
> In the machine to do thread cutting, connect mechanically between the spindle and position coder by the gear ratio=1:1. And these parameters set the No.3721=0, No.3722=0.
> (It is the setting for the gear ratio 1:1.)
> 2 Parameter No.3721 and No.3722 can not be used for Cs contour control. If Cs contour control was done by setting the any value on these parameters, spindle can’t move as commanded. In the machine to do Cs contour control, connect mechanically between the spindle and position coder by the gear ratio = 1:1. And these parameters set the No.3721=0, No.3722=0. (It is the setting for the gear ratio 1:1.)

3738	Spindle name 2 of each spindle
3739	Spindle name 3 of each spindle

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 48 to 57, 65 to 90

The command for a spindle is basically "S".
When all conditions below are satisfied, however, an extended spindle name can be used. An extended spindle name consists of up to three characters starting with "S" as the first spindle name. Thus, a command for a spindle can be specified.

- The serial (analog) spindle function is enabled.
- The multi-spindle control is enabled.
- Bit 0 (EEA) of parameter No. 1000 is set to 1.
- Bit 3 (MPP) of parameter No. 3703 is set to 1.
- Bit 1 (ESN) of parameter No. 3798 is set to 1.
- Bit 4 (GTT) of parameter No. 3706 is set to 1. (M series only)

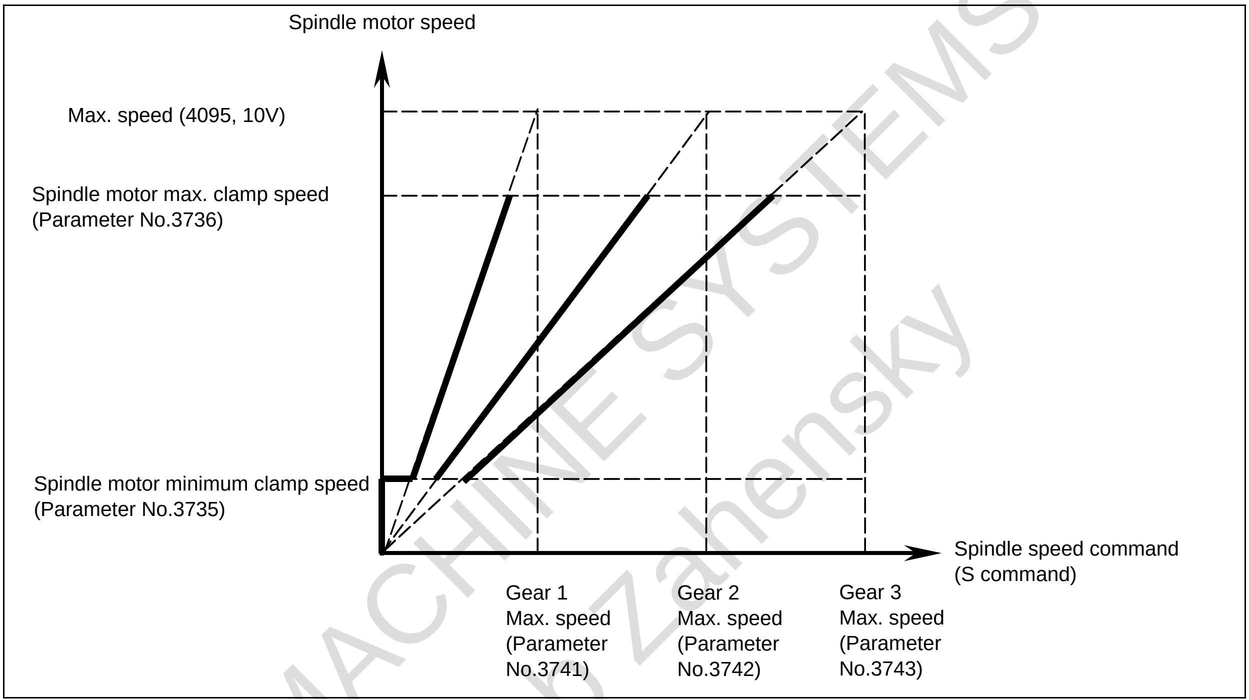
As spindle name 2 (parameter No. 3738) and spindle name 3 (parameter No. 3739), ASCII codes from 0 to 9 and A to Z can be arbitrary set. However, before spindle name 3 for a spindle can be valid, spindle name 2 must be set for the spindle. Moreover, when a character from 0 to 9 is set as spindle name 2, do not set a character from A to Z as spindle name 3.

> **NOTE**
> 1 When an extended spindle name is used, a subscript (for a main spindle (parameter No. 3718)) and a subscript (for a sub-spindle (parameter No. 3719)) are unusable.
> 2 When the custom macro function is enabled, the same extended spindle name as a reserved word must not be used. Such an extended spindle name is regarded as a reserved word.

3741	Maximum spindle speed for gear 1
3742	Maximum spindle speed for gear 2
3743	Maximum spindle speed for gear 3
3744	Maximum spindle speed for gear 4

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

Set the maximum spindle speed corresponding to each gear.



3772	Maximum spindle speed

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

This parameter sets the maximum spindle speed.
When a command specifying a speed exceeding the maximum speed of the spindle is specified , or the speed of the spindle exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

> ⚠ **CAUTION**
> 1 When 0 is set in this parameter, the speed of the spindle is not clamped.
> 2 When spindle speed command control is applied using the PMC, this parameter has no effect, and the spindle speed is not clamped.

> **NOTE**
> 1 For M series, this parameter is valid if the function of constant surface speed control is enabled(bit 0(SSC) of parameter No.8133 is 1).
> 2 When the constant surface speed control is enabled, the spindle speed is clamped at the maximum speed, regardless of whether the G96 mode or G97 mode is specified.

3775	Default P command value for spindle selection in multi-spindle control

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 32767

When bit 3 (MPP) of parameter No. 3703 is set to 1 and bit 2 (MPA) of parameter No. 3706 is set to 1 in multi-spindle control, set a default P command value applicable if S_P_ is not specified even once after power-up.

3781	P code for selecting the spindle in multi-spindle control

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767

If bit 3 (MPP) of parameter No. 3703 is set to 1, set the P code to select each spindle under multi-spindle control. Specify the P code in a block containing the S command.

[Example] If the P code value for selecting the second spindle is set to 2,
S1000 P2;
causes the second spindle to rotate at S1000.

> **NOTE**
> 1 This parameter is valid if bit 3 (MPP) of parameter No. 3703 is set to 1.
> 2 If this parameter is set to 0, the corresponding spindle cannot be selected by a P code.
> 3 Under multi-path control, the P code specified here is valid for each path.
> For instance, if the P code to select the first spindle of path 2 is set to 21, specifying S1000 P21; in path 1 causes the first spindle of path 2 to be rotated at S1000.
> 4 Identical P code values cannot be used for different spindles. (Identical P code values cannot be used even if the paths are different.)
> 5 When this parameter is used (when bit 3 (MPP) of parameter No. 3703 is set to 1), the spindle command selection signal is invalid.
> 6 To use this parameter, enable the multi-spindle control function(bit 1(MSP) of parameter No.8133 is 1).

	#7	#6	#5	#4	#3	#2	#1	#0
3786						MPF		

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#2 MPF When M03/M04/M05 is commanded by "M_P_" format in multi-spindle control, multi-spindle address P signal <Fn160-Fn161> is:
0: Not output.
1: Output.

> **NOTE**
> This parameter is valid when bit 3 (MPP) of parameter No.3703 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
3798							ESN	

[Input type] Parameter input
[Data type] Bit

#1 ESN When the multi-spindle control is enabled and bit 3 (MPP) of parameter No. 3703 is set to 1, a spindle is specified in a program by using:
0: P command.
1: Extended spindle name.

A spindle to be specified is selected as follows:

Bit 1 (ESN) of parameter No. 3798	Bit 3 (MPP) of parameter No. 3703	Selection method
0	0	Signal selection
0	1	P command (S_P_)
1	0	Signal selection
1	1	Extended spindle name or P command (S_P_)

> **NOTE**
> This parameter is valid when bit 0 (EEA) of parameter No. 1000 is set to 1.
> When setting this parameter to 1, set also parameter No. 3738 and No. 3739 properly.

11090	Path number with which the rotation of each spindle is specified

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 10

When a path is specified for spindle commands, this parameter sets a path number with which the rotation of a spindle can be specified.
0: Spindle commands can be issued from all paths.

1 to 10: Spindle commands can be issued from a set path.

NOTE

1. This parameter is valid when spindle command path specification signal SPSP<Gn536.7> is set to "1".
2. If the setting is illegal, an alarm PS5305, “ILLEGAL SPINDLE NUMBER” is issued when a spindle command is issued from any one of the paths.
3. This setting does not apply to spindle commands using the spindle select signals SWS1 to SWS4<Gn027.0 to Gn027.2, Gn026.3>.

Caution

⚠ **CAUTION**

1. If the primary spindle stop signal *SSTP<Gn029.6> for stopping all selected (SWS1 to SWS4) spindles’ rotation is cleared, the speed command is restored. A spindle not selected by SWS1 to SWS4 and rotating at its previous speed, which is stopped using its respective command *SSTP1 to *SSTP4, cannot be restored to that speed when the signal is cleared.
2. While a function using a position coder feedback such as feed per revolution and a threading command is being executed, avoid changing the selection of a position coder by using position coder selection signals PC2SLC<Gn028.7> , PC3SLC<Gn026.0> , and PC4SLC<Gn026.1> .
3. Type A multi-spindle control differs from Type B in the relationship between the SWS1 and SIND signals for the first spindle. In Type A, SIND functions only when SWS1 is set to "1". In Type B, SIND functions whether SWS1 is "1" or "0"; each spindle is selected by either of its respective SWS1 or SIND signals being set to "1".

Note

NOTE

1. The spindle orientation signal SOR<Gn029.5>, spindle speed override signals SOV0 to SOV7<Gn030>, and spindle stop signal *SSTP <Gn029.6> only function for the selected spindle.
2. The multi-spindle control allows multiple position coder interfaces to be used. But the number of actual speed indications on the CNC screen does not change. The speed based on the feedback information of the selected position coder is displayed.
3. An SOR command has priority over S commands and SIND-based rotation control from the PMC, and will cause all selected spindles to perform orientation rotation.

Alarm and message

Number	Message	Description
PS0333	TOO MANY SPINDLE COMMANDS	Multiple spindle commands could be found in the same block in using an expansion spindle name. Only one spindle could be commanded in the same block.

Number	Message	Description
PS5305	ILLEGAL SPINDLE NUMBER	In a spindle select function by address P for a multiple spindle control, 1 Address P is not specified. 2 Parameter No. 3781 is not specified to the spindle to be selected. 3 An illegal G code which cannot be commanded with an S_P_; command is specified. 4 The multi-spindle control cannot be used because the bit 1 (EMS) of parameter No. 3702 is 1. 5 The spindle amplifier number of each spindle is not set in parameter No. 3717. 6 A prohibited command for a spindle was issued (parameter No. 11090). 7 An invalid value is set in parameter No. 11090.
SP1202	SPINDLE SELECT ERROR	In the multi-spindle control, the spindle number other than the valid spindle number was selected by a position coder select signal. An attempt was made to select the spindle number of the system having no valid spindle.

10.13 RIGID TAPPING

Overview

In a tapping cycle (Machining center system: G84/G74, Lathe system: G84/G88), synchronous control is applied to the tapping operation of a tapping axis and the operation of the spindle.
This capability eliminates the need to use a tool such as a float tapper, thus enabling higher-speed, higher-precision tapping.
Whether a tapping cycle is an ordinary tapping cycle or rigid tapping cycle is determined by the miscellaneous function code for rigid tapping M29. (A different M code can be used by setting the parameters accordingly, but M29 is used in the description given here.)
By setting the parameters, G codes for tapping cycles can be changed to G codes for rigid tapping only. In this case, the CNC specifies M29 internally.
To perform rigid tapping, the following must be added to the ordinary connections:

- Connection of a position coder to the spindles
- Addition of a sequence to the PMC
- Setting of related parameters

To avoid duplicate descriptions, assume the following unless noted otherwise:

G code for a tapping cycle	
Machining center system	**Lathe system**
G84/G74	G84/G88

Gear selection method	
Machining center system	**Lathe system**
M-type or T-type gear selection method	T-type gear selection method only

Parameters used according to the number of gear stages
No.5221 to No.5224, No.5231 to No.5234, No.5241 to No.5244, No.5261 to No.5264, No.5271 to No.5274, No.5281 to No.5284, No.5291 to No.5294, No.5321 to No.5324,etc.

> ⚠ **CAUTION**
> 1 In the text of this section, a description of up to four-step gears is provided. Gears of four steps are used with the lathe system, but gears of up to only three steps are used with the machining center system.
> 2 When M-type gear selection is used for the machining center system, the maximum spindle speed for rigid tapping (specified with parameters Nos. 5241 to 5243) must also be set for parameter No. 5243 regardless of the number of gear steps. (For a system having a single gear step, set the same value as that of parameter No. 5241 for parameter No. 5243. For a system having two gear steps, set the same value as that of parameter No. 5242 for parameter No. 5243.)

The descriptions given in this section (such as spindle gear switching and M-type/T-type) are based on the explanation given in Section "SPINDLE SPEED CONTROL."
Refer to Section "SPINDLE SPEED CONTROL" as necessary.

- Specification of machining center system/lathe system

The differences in the specifications for rigid tapping for the machining center system and lathe system are described below.

M

Rigid tapping of machining center system

The tapping cycle G84 and the reverse tapping cycle G74 can be used to specify machining center system rigid tapping.
A drilling axis can be arbitrarily selected from the basic axes X, Y, Z, as well as axes parallel to the basic axes, by setting the corresponding parameters accordingly (bit 0 (FXY) of parameter No. 5101).
The spindle operations of G84 and G74 are reversed with respect to each other.

T

Rigid tapping of lathe system

The face tapping cycle G84 and the side tapping cycle G88 can be used to specify lathe system rigid tapping.
Depending on the rigid tapping command, rigid tapping can be performed along the Z-axis (when G84 is used) or the X-axis (when G88 is used).
A reverse tapping cycle, like that supported by machining center system, is not available.

See "Rigid Tapping with Spindle of Another Path" for explanations about rigid tapping between different paths (rigid tapping on a spindle for tool post 2 as directed by a command for tool post 1) in a multi-path system.

- Multi-spindle control

The relationships between rigid tapping and multi-spindle control are described below.

Without multi-spindle control

Rigid tapping can be performed with the first spindle only.

With multi-spindle control

Rigid tapping can be performed by selecting the second through fourth spindles in addition to the first spindle.

10.13.1 Connection Among Spindle, Spindle Motor, and Position Coder

As shown in the figures (Fig. 10.13.1 (a) and Fig. 10.13.1 (b)) below a gear ratio can be inserted between the spindle and spindle motor (n : m), and between the spindle and position coder (N : M).

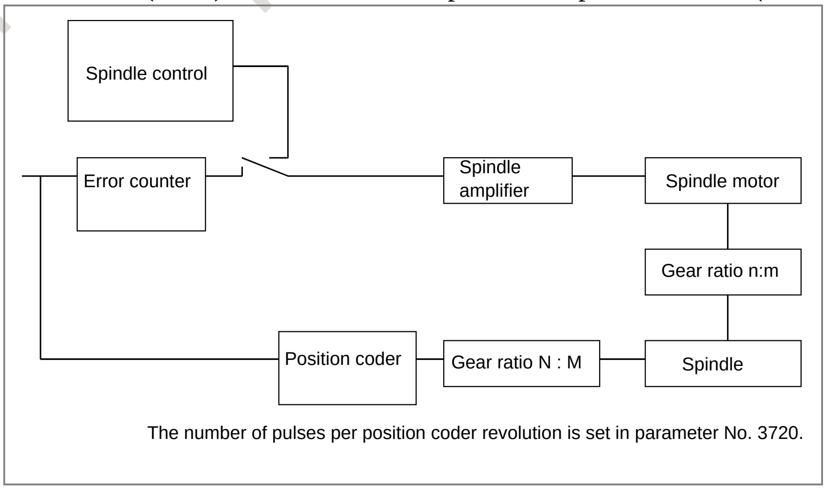


Fig. 10.13.1 (a) When a position coder is externally attached (on the spindle side)

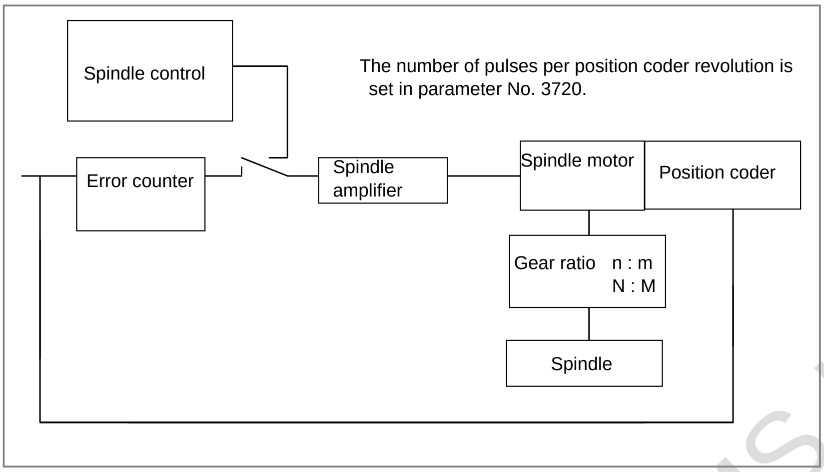


Fig. 10.13.1 (b) When a position coder is built into a spindle motor

(1) Gear between spindle and spindle motor

Up to three gear stages (machining center system) or four gear stages (lathe system) can be provided between the spindle and the spindle motor.

The gear ratio is arbitrary. The distance of spindle rotation per revolution of spindle motor is different, based on the gear ratio. The speed command to the spindle motor will be adjusted.

(2) Gear between spindle and position coder

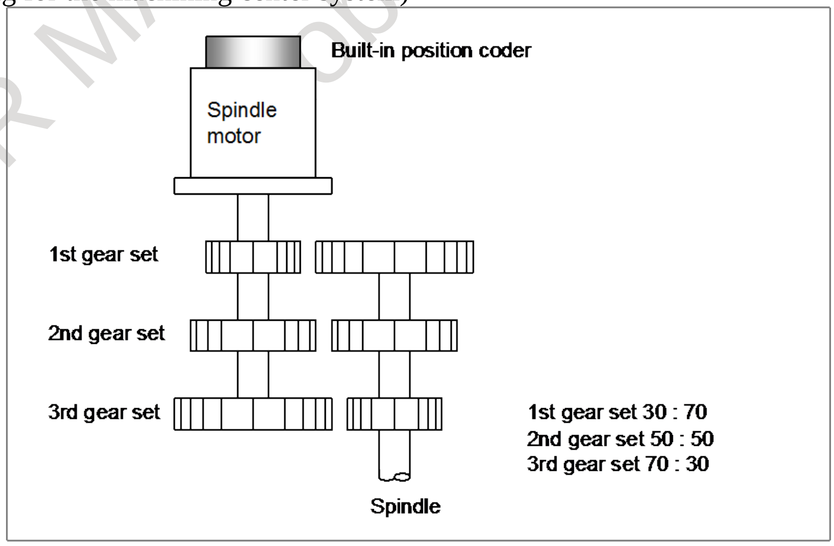
The position coder is used to detect the position of the spindle.

The gear ratio between the spindle and position coder is set in parameters Nos. 5221 to 5224 and 5231 to 5234.

When a position coder is externally attached, the gear ratio between the spindle and position coder is not changed by a gear change between the spindle motor and spindle. So, set the same number of teeth in parameters Nos. 5221 to 5224 and 5231 to 5234.

When a position coder is built into a spindle motor, the gear ratio between the spindle and position coder is changed by a gear change between the spindle motor and spindle. So, set the number of teeth for each gear in parameters Nos. 5221 to 5224 and 5231 to 5234.

Example of setting for the machining center system)



Parameter No.	Set value	Meaning
5221	70	Number of teeth of the 1st gear for the spindle side
5222	50	Number of teeth of the 2nd gear for the spindle side
5223	30	Number of teeth of the 3rd gear for the spindle side
5231	30	Number of teeth of the 1st gear for the position coder side

Parameter No.	Set value	Meaning
5232	50	Number of teeth of the 2nd gear for the position coder side
5233	70	Number of teeth of the 3rd gear for the position coder side

For the lathe system, up to four gear stages can be set.

(3) Rigid tapping and machines with multiple gears

When performing rigid tapping on a machine that has multiple gears, note the points below.

- M type gear selection method

M

The CNC determines whether gears need changing using the gear change specification mentioned in section “SPINDLE SPEED CONTROL”. If the gears need to be changed, the CNC generates the S function code read signal SF <F007.2> and gear selection signals GR1O, GR2O, and GR3O <F034.0-2> to notify the PMC. Change gears using the PMC, based on these signals.

- T type gear selection method

The CNC does not process gear changes. Using the PMC, determine whether gears need changing, and make the change if needed.
When the CNC has the S function code, it outputs signal SF <F007.2> and S function code signals S00 to S31 <F022-F025> to the PMC. (However, parameter No. 3705 and its related parameters need to be set for S code and SF signal output).
The PMC side finds a gear whose spindle speed range includes the spindle speed specified by an S function code. A selected gear is to be output onto the gear selection signals GR1 and GR2 (G028.1,2 for the first spindle, G029.0,1 for the second spindle, G029.2,3 for the third spindle, and G031.4,5 for the fourth spindle) for notification to the CNC. Regardless of the gear selection method, the clutch/gear selection signals CTH2 and CHT1 (G070.3,2 for the first spindle, G074.3,2 for the second spindle, G204.3,2 for the third spindle, and G266.3,2 for the fourth spindle) corresponding to a gear change are to be input for notification through the CNC to the serial spindle control unit.

Changing gears during rigid tapping requires a different process from that for gear changes during normal machining. As described above, changing gears conforms to the gear change specifications mentioned in section "SPINDLE SPEED CONTROL" when the M type gear selection method has been selected. With the T type gear selection method, changing gears conforms to the logic programmed in the PMC.
Regardless of the option's selection, if the range in which the spindle speed specified by the S function code does not correspond to the currently selected gear, the gears are changed.
The Table 10.13.1 (a) lists the spindle speed ranges for each gear during normal machining (assuming no machine restrictions) and rigid tapping:

Table 10.13.1 (a)

Gear	Spindle speed range (normal machining)	
	Lower limit	Upper limit
Low-speed gear	1 revolution	Maximum low-speed gear speed $= \dfrac{\text{Maximum spindle motor speed} \times \text{L\%}}{\text{Low-speed gear ratio}}$
Medium speed gear	Maximum low-speed gear speed + 1 revolution	Maximum medium-speed gear speed $= \dfrac{\text{Maximum spindle motor speed} \times \text{L\%}}{\text{Medium speed gear ratio}}$
High-speed gear	Maximum medium-speed gear speed + 1 revolution	Maximum high-speed gear speed $= \dfrac{\text{Maximum spindle motor speed} \times \text{L\%}}{\text{High-speed gear ratio}}$

> **NOTE**
> Table 10.13.1 (a) shows an example of three gears. L% indicates a spindle motor protection constant (up to 100). L can be specified for each gear using method B for changing in M type gear selection method (bit 2 (SGB) of parameter No. 3705 is set to 1).

The spindle speed range of each gear in rigid tapping must satisfy the Table 10.13.1 (b).

Table 10.13.1 (b)

Gear	Spindle speed range (during rigid tapping)	
	Lower limit	Upper limit
Low-speed gear	1 revolution	Maximum low-speed gear speed $= \dfrac{\text{Basic spindle motor speed} + \alpha}{\text{Low-speed gear ratio}}$
Medium speed gear	Maximum low-speed gear speed + 1 revolution	Maximum medium-speed gear speed $= \dfrac{\text{Basic spindle motor speed} + \alpha}{\text{Medium-speed gear ratio}}$
High-speed gear	Maximum medium-speed gear speed + 1 revolution	Maximum high-speed gear speed $= \dfrac{\text{Basic spindle motor speed} + \alpha}{\text{High-speed gear ratio}}$

> **NOTE**
> This table show an example of three gears. For the basic spindle motor speed, refer to the spindle motor description manual. "+α" means that the spindle motor speed may slightly exceed the basic spindle motor speed.

If the M type gear selection method is used, use gear change method B (bit 3 (SGT) of parameter No. 3705 is set to 1) in the tapping cycle to specify the following:
The table above shows the maximum low-speed gear speed during rigid tapping for gear 1 to gear 2 change point (parameter No. 3761). The table above shows the maximum medium-speed gear speed during rigid tapping for gear 2 to gear 3 change point (parameter No. 3762).
If the T type gear selection method is used, add the rigid tapping logic to the logic programmed in the PMC. See Section "SPINDLE SPEED CONTROL" for details of the spindle gear change specifications.
The loop gain can be specified for each gear. In parameters Nos. 5281 to 5284, set the loop gain of each gear. Set a loop gain multiplier in parameters Nos. 5291 to 5294. Specify the time constant and the maximum spindle speed for each gear.
Use parameters Nos. 5261 to 5264 to specify the time constant.
Use parameters Nos. 5241 to 5244 to specify the maximum spindle speed.
For M type gear selection method, set the maximum spindle speed to parameter No. 5243, irrespective of the number of gear stages used.
Setting bit 2 (TDR) of parameter No. 5201 to 1 enables setting of the time constants used during extraction for each gear set. Specify the extraction time constant for each gear in parameters Nos. 5271 to 5274.
It is recommended that a value of about 1:8 to 8:1 be actually used as the gear ratio between the spindle and position coder.

10.13.2 Rigid Tapping Specification

- Feedrate

In rigid tapping mode, the drilling axis is fed at a rate specified by F. The spindle speed is specified by S 360 [deg/min].
A detailed description of commands for feed per minute and feed per revolution is provided later.

- Acceleration/deceleration after interpolation

Linear acceleration/deceleration or bell-shaped acceleration/deceleration can be applied.
A detailed description is provided later.

- Look-ahead acceleration/deceleration before interpolation

Look-ahead acceleration/deceleration before interpolation is disabled.

- Override

Various types of overrides are disabled. However, the following overrides can be enabled by setting parameters:
- Extraction override
- Override signal

A detailed description is provided later.

- Dry run

Dry run is valid for G84/G74 (for machining center system) or G84/G88 (for lathe system). When the dry run is applied to the drilling axis speed of G84/G74 or G84/G88, tapping is performed. The spindle speed will match the dry run speed.

- Machine lock

When G84/G74 (for machining center system) or G84/G88 (for lathe system) is executed with the machine locked, however the drilling axis and the spindle do not move.

- Reset

When a reset operation is performed during rigid tapping, rigid tapping is stopped. Rigid tapping in-progress signal RTAP<Fn076.3> and spindle enable signal ENB<Fn001.4>, ENB2<Fn038.2>, ENB3<Fn038.3>, ENB4<Fn039.1> are set to "0". Set rigid tapping signal RGTAP<Gn061.0> to "0" when falling edge of spindle enable signal ENB<Fn001.4>, ENB2<Fn038.2>, ENB3<Fn038.3>, ENB4<Fn039.1> is detected. When a reset operation is performed, G code is set as Table 10.13.2 according to bit 5 (CLR) of parameter No.3402, bit 1 (C09) of parameter No.3407 (machining center system), and bit 2 (C10) of parameter No.3407 (lathe system). In restarting operation with G84/G74 (machining center system), G84/G88 (lathe system), command G80 if necessary.

Table 10.13.2

CLR (bit 5 of parameter No.3402)	C09 (bit 1 of parameter No.3407) (machining center system) C10 (bit 2 of parameter No.3407) (lathe system)	G code
0	-	G84/G74 (machining center system) G84/G88 (lathe system)
1	0	G80
1	1	G84/G74 (machining center system) G84/G88 (lathe system)

- Feed hold and single block

Feed hold and single block are disabled in rigid tapping. When bit 6 (FHD) of parameter No.5200 is 1, feed hold and single block are enabled.

- Feed hold
  - Bit 6 (FHD) of parameter No.5200 = 0
    If feed hold is applied between Motion 3 and Motion 5 described at Section "Command Format", the feed hold lamp turns on immediately, but the tool decelerates and stops after moving to Motion 6. If feed hold is applied during Motion 1, Motion 2, and Motion 6, CNC becomes feed hold state and the tool decelerates and stops.
  - Bit 6 (FHD) of parameter No.5200 = 1
    If feed hold is applied between Motion 1 and Motion 6, CNC becomes feed hold state and the tool decelerates and stops.

- Single block
  - Bit 6 (FHD) of parameter No.5200 = 0
    Single block stop points are the end points of Motion 1, Motion 2, and Motion 6.
  - Bit 6 (FHD) of parameter No.5200 = 1
    Single block stop points are the end points of each Motion.

- Manual feed

When performing rigid tapping by using manual handle feed, see the description of the “RIGID TAPPING BY MANUAL HANDLE”.
Rigid tapping cannot be used with other types of manual feed.

- Backlash compensation

In rigid tapping mode, the backlash is compensated for the lost motion at forward and reverse spindle rotations. Set it using parameters Nos. 5321 to 5324. The normal backlash compensation is inserted on the drilling axis.

- C-axis clamp/unclamp

T

An M code can be specified to clamp/unclamp the C-axis mechanically in rigid tapping. By adding an M code for clamping to a block specifying G84/G88 (lathe system), both M codes are output. A description of timing is provided later.
An M code for clamping is to be set in parameter No. 5110. An M code for unclamping assumes the value set in parameter No. 5110 + 1.

No.5110	
0	Other than 0
No M code is output.	M code is output.

10.13.3 Commands for Feed per Minute and Feed per Revolution

In rigid tapping commands for feed per minute and feed per revolution can be specified.

(Example)
The example below specifies rigid tapping in feed per minute mode for cutting a thread with a lead of 1 mm at a spindle speed of 1,000 min^{-1}. (In feed per minute mode, F/S determines the thread lead.)

```
O0002 ;
G94 ;
```

```
  :
M29 S1000 ;
G84 X50. Y30. Z-100. R-20. F1000 ;
  :
G80 ;
```

The example below specifies rigid tapping in feed per revolution mode for cutting the same thread at the same spindle speed as above. (In feed per rotation mode, F determines the thread lead.)

```
O0001 ;
G95 ;
  :
M29 S1000 ;
G84 X50. Y30. Z-100. R-20. F1. ;
  :
G80 ;
```

> **NOTE**
> Even in the feed per revolution mode, pulses distributed for the drilling axis are converted to a command for feed per minute. Thus, feed per rotation mode does not strictly implement feed per rotation. Accordingly, even if the spindle stops for some reason, the drilling axis does not stop.

10.13.4 Acceleration/Deceleration after Interpolation

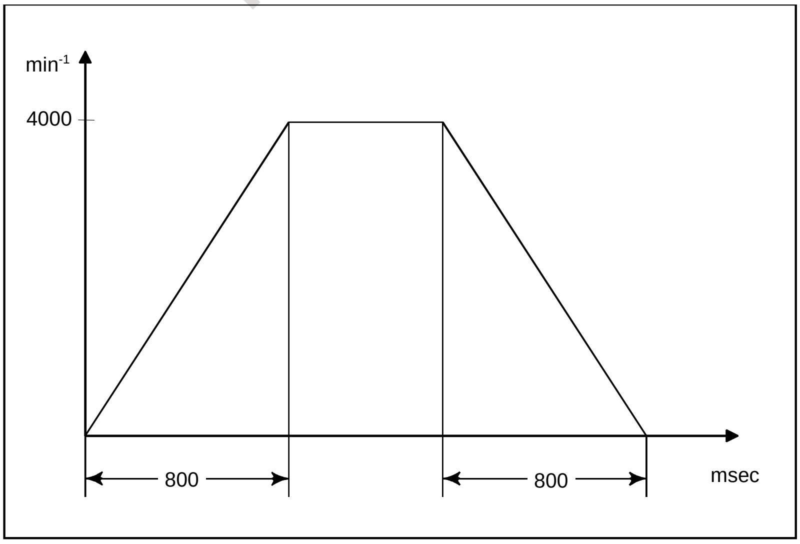
Linear acceleration/deceleration or bell-shaped acceleration/ deceleration can be applied.
When bit 2 (TDR) of parameter No. 5201 is set to 1, time constants for cutting and extraction can be set in separate parameters (Parameters Nos. 5271 to 5274).

- Linear acceleration/deceleration

By setting bit 5 (RBL) of parameter No. 5203 to 0, linear acceleration/deceleration of constant acceleration type can be applied.
In parameters Nos. 5261 to 5264, set a time required to reach a maximum spindle speed.
The actual time constant is the ratio between a maximum spindle speed and S command value.

Example)

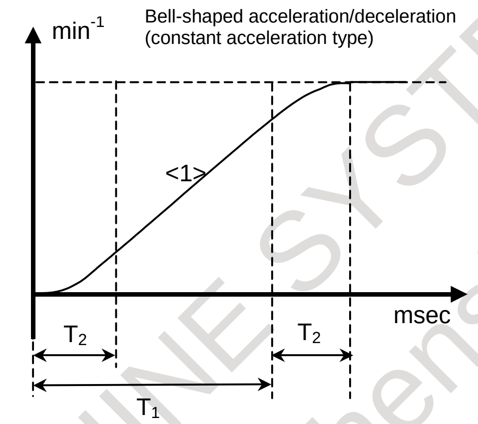
Suppose that the parameters are set as follows: time constant TC = 800 msec and maximum spindle speed S = 4000 min^{-1}.



When S2000 is specified, the acceleration/deceleration time is 400 msec. When S1000 is specified, the acceleration/deceleration time is 200 msec. This means that the acceleration rate in linear acceleration/deceleration at rigid tapping time is constant.

- Bell-shaped acceleration/deceleration

By setting bit 5 (RBL) of parameter No. 5203 to 1, bell-shaped acceleration/deceleration of constant acceleration type can be applied.
In parameters Nos. 5261 to 5264, set a time required to reach a maximum spindle speed (linear portion).
In parameters Nos. 5365 to 5368, set a time for the curved portion.
The actual time constant for the linear portion is the ratio between a maximum spindle speed and S command value.
To the curved portion, not an S command value but a set value is applied.
This means that the actual time constant is [linear portion value obtained by proportional calculation + value for curved portion].



Set time constants T_1 and T_2 in parameters.
(Acceleration rate of linear portion <1> is constant from settings of maximum spindle speed and T_1.)

NOTE

1. Even when bit 2 (TDR) of parameter No. 5201 is set to 1, the values set in parameters Nos. 5365 to 5368 are used for cutting and extraction in the curved portions.
2. When 0 is set in parameters Nos. 5365 to 5368, linear acceleration/deceleration is used.
3. In 3-dimensional rigid tapping, bell-shaped acceleration/deceleration is disabled, and linear acceleration/deceleration is used.

10.13.5 Override

Various types of overrides are disabled. However, the following overrides can be enabled by setting parameters:

- Extraction override
- Override signal

(1) Extraction override

For extraction override, a parameter-set fixed override value or programmed override value can be used at the time of extraction (including extraction operation at peck drilling/high-speed peck drilling).

- Parameter specification

Set bit 4 (DOV) of parameter No. 5200 to 1, and set an override value in parameter No. 5211.
An override value from 0% to 200% can be set in steps of 1%. When bit 3 (OVU) of parameter No. 5201 is set to 1, an override value from 0% to 2000% can be set in steps of 10%.

- Program specification

When bit 4 (DOV) of parameter No. 5200 and bit 4(OV3) of parameter No. 5201 are set to 1, a spindle speed to be used at extraction time can be programmed.
Using address J in a block specifying rigid tapping, specify a spindle speed to be used at extraction time.

Example)
When S = 1000 min^{-1} for cutting and S = 2000 min^{-1} for extraction

```
   :
M29 S1000 ;
G84 Z-100. F1000. J2000 ;
   :
```

The following calculation is made for conversion to an actual override value:

$$\text{Override value (\%)} = \frac{\text{Spindle speed for extraction (}J\text{ command)}}{\text{Spindle speed (}S\text{ command)}} \times 100$$

So, the spindle speed at extraction time may not match the speed specified by address J. If an override value not within the range 100% to 200% is specified, the specification of 100% is assumed.

The table below summarizes the setting of an extraction override value by parameter specification / program specification.

Command \ Parameter setting		DOV=1, OV3=1	DOV=1, OV3=0	DOV=0
Spindle speed for extraction is specified by address J.	**Within the range 100% to 200%**	Program specification	Parameter No. 5211	100%
	Not within the range 100% to 200%	100%		
Spindle speed for extraction is not specified by address J.		Parameter No. 5211		

NOTE

1 When specifying address J, do not use the decimal point.
If the decimal point is used, the specified value is handled as described below.
Example) When the increment system for the reference axis is IS-B
- When calculator-type decimal point input is not used
The specified value is converted to a value considering the least input increment.
"J200.0" is converted to 200000 min^{-1}.
- When calculator-type decimal point input is used
The specified value is converted to a value with the decimal places discarded.
"J200.0" is converted to 200 min^{-1}.

2 When specifying address J, do not use the minus sign.
When the minus sign is used, the specification of a value not within the range 100% to 200% is assumed.

3 Find a maximum allowable extraction override value according to the expression below so that the spindle speed multiplied by an extraction override value does not exceed the maximum speed (parameters Nos. 5241 to 5244) of the gear in use. For this reason, a mismatch with the maximum spindle speed can occur, depending on the override value.

$$\text{Maximum override value (\%)} = \frac{\text{Maximum spindle speed (parameter setting)}}{\text{Spindle speed (}S\text{ command)}} \times 100$$

> **NOTE**
> 4 Address J for specifying a spindle speed for extraction, when specified in the rigid tapping mode, remains valid until the canned cycle is canceled.

(2) Override signals

When bit 4 (OVS) of parameter No. 5203 is set to 1, a cutting/extraction operation in rigid tapping can be overridden as described below.

- Override is applied using the feedrate override signal *FV0 to *FV7 <Gn012>.
  (When the second feedrate override signal is valid, the second feedrate override is applied to the feedrate to which the feedrate override is already applied.
- Override is canceled using the override cancel signal OVC <Gn006.4>.
  The relationship between this function and overriding each operation is described below.
- At the time of cutting
  When the override cancel signal = 0
    Value specified by the override signal
  When the override cancel signal = 1
    100%
- At the time of extraction
  When the override cancel signal = 0
    Value specified by the override signal
  When the override cancel signal = 1
    When extraction override is disabled
      100%
    When extraction override is enabled
      Value specified by an extraction override value

> **NOTE**
> Find a maximum allowable override value according to the expression below so that the spindle speed multiplied by an override value does not exceed the maximum speed (parameters Nos. 5241 to 5244) of the gear in use. For this reason, a mismatch with the maximum spindle speed can occur, depending on the override value.
>
> $$\text{Maximum override value (\%)} = \frac{\text{Maximum spindle speed (parameter setting)}}{\text{Spindle speed (}S\text{ command)}} \times 100$$

10.13.6 Reference Position Return

For serial spindles, setting bit 0 (ORI) of parameter No. 5202 to 1 enables a reference position return to be made for a spindle at the start of rigid tapping.
When a movement is made to a R point in the rigid tapping mode, the reference position return direction is determined on the serial interface spindle side for reference position return. With the grid shift function, the reference position can be shifted within the range 0° to 360°.
Set a reference position return direction in bit 4 (RETSV) of parameter No. 4000. Set a grid shift amount in parameter No. 4073.

> **NOTE**
> 1 A feedrate for reference position return is determined from the position where the tool is stopped when the rigid tapping mode is set, and the amount of travel found from the reference position. Accordingly, no fixed speed is applied.
> 2 When the gear ratio between the spindle and position coder is not $1:2^n$ (n: integer other than 0), the tool may not be able to return to the same reference position.

10.13.7 FANUC Series 15 Program Format Command

M

When bit 1 (FCV) of parameter No. 0001 is set to 1, rigid tapping can be specified in the FANUC Series 15 command format.

```
(1)   :
     Sxxxx ;
     G84.2/G84.3 X_ Y_ Z_ R_ P_ F_ L_ ;
      :
(2)   :
     G84.2/G84.3 X_ Y_ Z_ R_ P_ F_ L_ Sxxxx ;
      :
```

G84.2	: Rigid tapping cycle
G84.3	: Rigid reverse tapping cycle
X,Y	: Tapping position
Z	: Position of the bottom of a hole
R	: Position of a R point
P	: Dwell time at the bottom of a hole and at R point return time
F	: Cutting feedrate
L	: Number of repeats
S	: Spindle speed

> **NOTE**
> This function enables the rigid tapping function to be specified in the FANUC Series 15 command format without changing the rigid tapping sequence (such as handling with the PMC).

T

When bit 1 (FCV) of parameter No. 0001 is set to 1 and bit 3 (F16) of parameter No. 5102 is set to 0, rigid tapping can be specified in the FANUC Series 15 command format.

```
(1)  :
     Sxxxx ;
     G84.2 X_ C_ Z_ R_ P_ F_ L_ ;
     :
(2)  :
     G84.2 X_ C_ Z_ R_ P_ F_ L_ S_ ;
     :
```

G84.2	: Rigid tapping cycle G code
X,C	: Tapping position
Z	: Position of the bottom of a hole
R	: Position of a R point
P	: Dwell time at the bottom of a hole and at R point return time
F	: Cutting feedrate
L	: Number of repeats
S	: Spindle speed

The differences from the FS16 command format are as follows:

- No G code can make a distinction between a face tapping cycle and side tapping cycle. A tapping axis is determined by plane selection (G17/G18/G19). Select a plane that matches a face tapping cycle/side tapping cycle.

Plane select	Drilling axis
G17: Xp-Yp plane	Zp
G18: Zp-Xp plane	Yp
G19: Yp-Zp plane	Xp

Xp : X axis or its parallel axis
Yp : Y axis or its parallel axis
Zp : Z axis or its parallel axis

NOTE
With bit 0 (FXY) of parameter No. 5101, the Z-axis can be used as the tapping axis at all times. If FXY is set to 0, the Z-axis is used as the tapping axis at all times.

- The M code for C-axis clamping cannot be specified.
- The specification of a R point depends on the G code system and parameter setting as described below.
  1. When the bit 3 (F16) of parameter No. 5102 is set 0.

Bit 6 (RAB) of parameter No. 5102		
0	1	
Incremental command at all times	G code system	
	A	B,C
	Absolute command at all times	Follows G90/G91.

Bit 7 (RDI) of parameter No. 5102	
0	1
Diameter command at all times	Follows the diameter specification/radius specification of the tapping axis.

  2. When the bit 3 (F16) of parameter No. 5102 is set 1.
     An incremental command based on radius specification is used, regardless of bit 6 (RAB) and bit 7 (RDI) of parameter No. 5102.

NOTE
This function enables the rigid tapping function to be specified in the FANUC Series 15 command format without changing the rigid tapping sequence (such as handling with the PMC).

10.13.8 Multi-Spindle Control

When multi-spindle control is used, rigid tapping can be performed by selecting spindles from the first to fourth spindles. A spindle can be selected as described below (Table 10.13.8) by parameter setting.

Table 10.13.8

		Machining center system	Lathe system	
			Bit 7 (SRS) of parameter No. 5200	
			0	1
Bit 3 (MPP) of parameter No. 3703	0	Selected using the spindle selection signals SWS1 to SWS4	Selected using the spindle selection signals SWS1 to SWS4	Selected using the rigid tapping spindle selection signals RGTSP1 to RGTSP4
	1	Selected using an NC command (address P)		

For details of the method of selection using the spindle selection signals/rigid tapping spindle selection signals, see the descriptions of signals provided later.
For details of the method of selection using an NC command (address P), see "MULTI-SPINDLE CONTROL" .

10.13.9 3-dimensional Rigid Tapping

By performing rigid tapping in 3-dimensional coordinate conversion mode or Tilted working plane indexing mode, the tapping operation can be directed to the angular direction specified by 3-dimensional conversion command or Tilted working plane indexing command.
Be sure to use this function with 3-dimensional coordinate conversion or Tilted working plane indexing.

10.13.10 Rigid Tapping with Spindle of Another Path

Overview

In multi-path system, rigid tapping with spindle of other path different from specified path can be executed by path spindle control function. The spindle which executes rigid tapping is selected according to the spindle command selection signals.
If multi-spindle control option is provided, not only the path but also the spindle executed with rigid tapping can be selected.

Specifications

- Case in which no multi-spindle control is available

If bit 4 (SPR) of parameter No. 3703 is set to 1, using spindle command select signals, like the S command, makes it possible to specify a path from which rigid tapping command is to be used to control a spindle in a specific path.
The spindle command select signals are described below.
(Example)

2-path system: SLSPA<G063.2> (for a spindle in path 1) and SLSPB<G063.3> (for a spindle in path 2)
3-path system: SLSPA$^{\#1}$<G0063.2>, SLSPB$^{\#1}$<G0063.3> (for a spindle in path 1), SLSPA$^{\#2}$<G1063.2>, SLSPB$^{\#2}$<G1063.3> (for a spindle in path 2), SLSPA$^{\#3}$<G2063.2>, and SLSPB$^{\#3}$<G2063.3> (for a spindle in path 3)

- Case in which the multi-spindle control is available

- Signal-based command
  Setting bit 1 (EMS) of parameter No.3702 to 0 and bit 4 (SPR) of parameter No.3703 to 1 enables an arbitrary spindle in an arbitrary path to be used for rigid tapping.
  Before a rigid tapping command is issued, the spindle to be used for rigid tapping is selected using a signal.
- Address P-based command
  Setting bit 1 (EMS) of parameter No.3702 to 0 and bit 4 (SPR) of parameter No.3703 to 1 enables to use, for rigid tapping, a spindle connected to a path other than the one in which the command is issued. Setting bit 3 (MPP) of parameter No. 3703 to 1 enables a spindle to be selected for use in rigid tapping according to a program (P code).
  Adding address P to the S command for rigid tapping enables a spindle to be selected for use in rigid tapping.
  (Example: **M29 S1000 P22 ;**)
  It is necessary to set, in advance, parameter No. 3781 with relationships between P values and axes selected with specific P values.

  Refer to descriptions of “MULTI-SPINDLE CONTROL” in the related connection manual for explanations about how to use address P to select a spindle.

Example 1 Case in which no multi-spindle control is used

On a 2-path lathe with 2 spindles under 2-path control (bit 0 (2P2) of parameter No. 3703 is set to 1), the following signals are set up as stated when rigid tapping is carried out using the rigid tapping command (machining program) and servo section (Z $^{\#1}$ in the Fig. 10.13.10 (a)) in path 1 and the first spindle (SP1 $^{\#2}$ in the Fig. 10.13.10 (a)) connected to path 2.

<1> SLSPB<G63.3> ="1": The spindle connected to path 2 is controlled by commands in path 1.
<2> SLSPA<G63.2> = "1": The spindle connected to path 1 is controlled by commands in path 2.
Rigid tapping is carried out on the first spindle (SP1$^{\#2}$) in path 2.

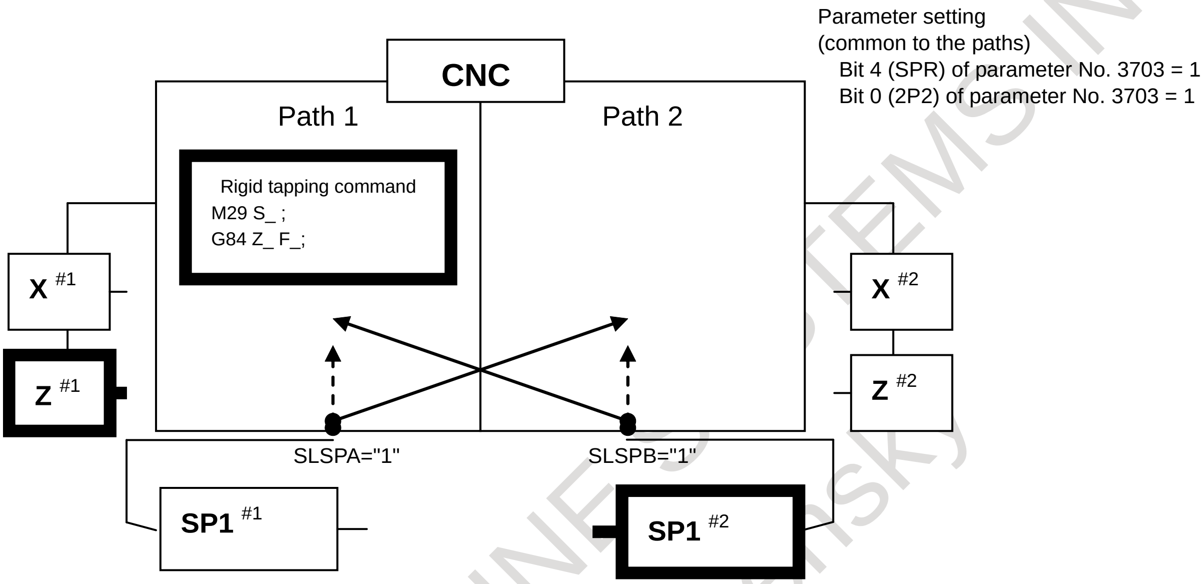


Fig. 10.13.10 (a)

(Supplement)

In the above example system configuration, the signal operation described in <2> is necessary to make each command correspond to the spindles selected by the commands on a one-to-one basis.
The way the spindle select signals are handled may vary according to the number of control paths in the CNC and the number of controlled axes (bits 0 (2P2) and 2 (MPM) of parameter No. 3703).

Example 2 Case in which multi-spindle control is used (signal-based command)

On a 2-path lathe with 4 spindles under 2-path control (bit 0 (2P2) of parameter No. 3703 is set to 1), the following signals are set up as stated beforehand when rigid tapping is carried out using the rigid tapping command (machining program) and servo section (Z $^{\#1}$ in the Fig. 10.13.10 (b)) in path 1 and the second spindle (SP2 $^{\#2}$ in the Fig. 10.13.10 (b)) connected to path 2.

<1> SLSPB<G63.3> = "1": The spindle connected to path 2 is controlled by commands in path 1.
<2> SLSPA<G63.2> = "1": The spindle connected to path 1 is controlled by commands in path 2.
<3> SWS1$^{\#2}$<G1027.0> ＝"0"
<4> SWS2$^{\#2}$<G1027.1> ＝"1": Rigid tapping is carried out on the second spindle (SP2$^{\#2}$) in path 2.

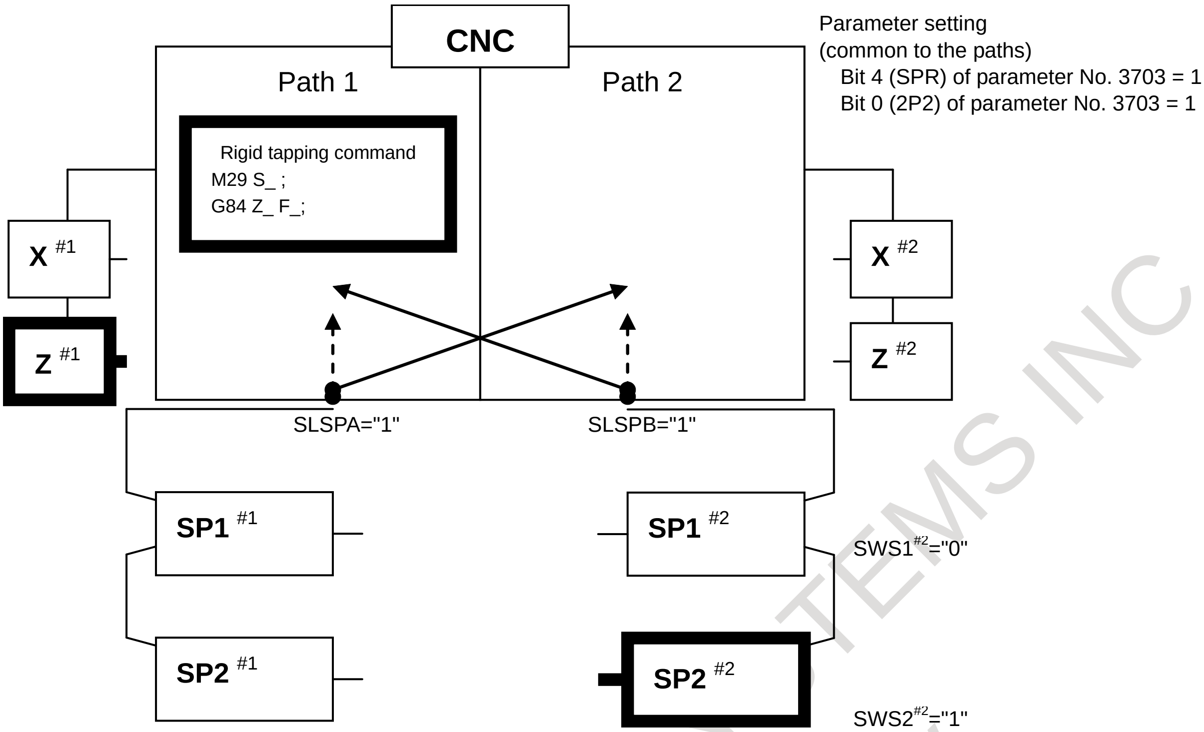


Fig. 10.13.10 (b)

(Supplement)

In the above example system configuration, the signal operation described in <2> is necessary to make each command correspond to the spindles selected by the commands on a one-to-one basis.

The way the spindle select signals are handled may vary according to the number of control paths in the CNC and the number of controlled axes (bits 0 (2P2) and 2 (MPM) of parameter No. 3703).

Example 3 Case in which multi-spindle control is used (address P-based command)

On a 2-path lathe with 4 spindles under 2-path control (bit 0 (2P2) of parameter No. 3703 is set to 1), before a rigid tapping command is issued, the following parameters are set up as stated if rigid tapping is to be carried out using the rigid tapping command (machining program) and servo section (Z $^{\#1}$ in the Fig. 10.13.10 (c)) in path 1 and the second spindle (SP2 $^{\#2}$ in the Fig. 10.13.10 (c)) connected to path 2.

No.3781 $^{\#1}$ S1 (Path 1: P value for selecting the first spindle) = 11
No.3781 $^{\#1}$ S2 (Path 1: P value for selecting the second spindle) = 12
No.3781 $^{\#2}$ S3 (Path 2: P value for selecting the first spindle) = 21
No.3781 $^{\#2}$ S4 (Path 2: P value for selecting the second spindle) = 22

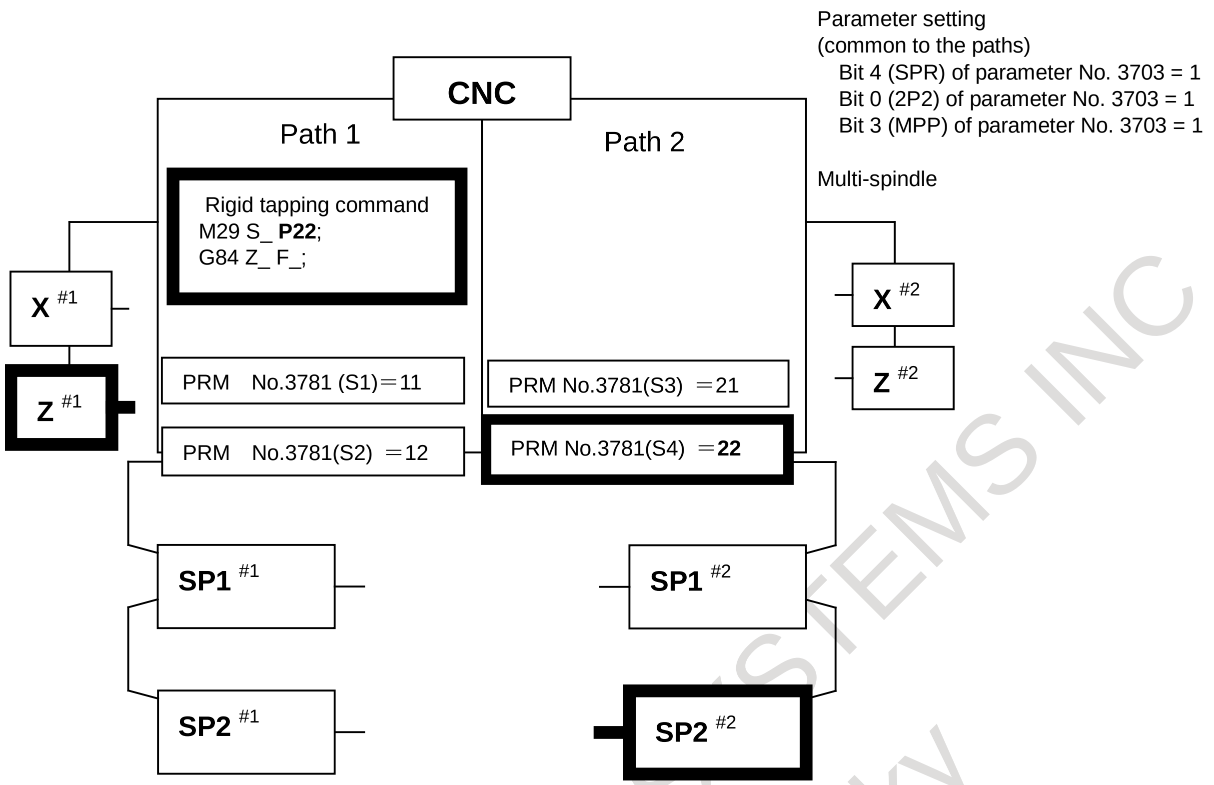


Fig. 10.13.10 (c)

The P code for selecting a spindle is added to the rigid tapping S command to execute a machining program. In this example, SP2 #2 must be controlled. So, the P code is used to issue a value (22) specified by parameter No. 3781 #2 (S4).

Restrictions

Observe the following restrictions.

- Paths issuing rigid tapping commands must correspond to spindles on which rigid tapping is carried out on a one-to-one basis.
- If this function is used to carry out rigid tapping on a spindle in a path other than a command-issuing path, the spindle must be put in the spindle speed control mode in advance.
- This function is unusable for servo motor-based rigid tapping.

The alarms below are raised if a restriction above is not observed, that is, if a condition stated below occurs.

Alarm PS0205 "RIGID MODE DI SIGNAL OFF"

- Rigid tapping commands in multiple paths attempted to carry out rigid tapping on the same spindle simultaneously.
- A rigid tapping command was issued to a spindle control axis with servo motor in a path other than the command issuing path.

Alarm SP0752 "SPINDLE MODE CHANGE ERROR"

- A spindle selected by a rigid tapping command was in a mode other than the spindle control mode.

The modes other than the spindle speed control mode include:

Cs contour axis
Spindle positioning
Spindle-synchronous
Rigid tapping (controlled by commands from another path)

T

inter-spindle polygon machining

[Example] 2-path system with each path having 2 spindles
(Rigid tapping is carried out on a spindle pointed to by an arrowhead)

	Path issuing commands		Spindle on which rigid tapping is carried out	MPP = 0	MPP = 1
Example 1	Path 1		First spindle	Available	Available
			Second spindle		
	Path 2		First spindle		
			Second spindle		
Example 2	Path 1		First spindle	Unavailable Alarm PS0205	Available
			Second spindle		
	Path 2		First spindle		
			Second spindle		
Example 3	Path 1		First spindle	Unavailable Alarm PS0205	Available
			Second spindle		
	Path 2		First spindle		
			Second spindle		
Example 4	Path 1		First spindle	Unavailable Alarm PS0205	Unavailable Alarm PS0205
			Second spindle		
	Path 2		First spindle		
			Second spindle		

If the rigid tapping spindle is not in the spindle speed control mode (for example, is in the Cs contour control mode), rigid tapping cannot be carried out in any of the above cases.

Notes

- This function is enabled when serial spindles are used. Issuing commands to an arbitrary spindle in a path other than a command issuing path requires the multi-spindle control to be enabled (bit 3 (MSP) of parameter No.8133 is 1).
- This function is unusable for rigid tapping with servo motor.
- If rigid tapping is carried out on a spindle in a path other than a command issuing path, the ENB signal is also shifted to the path to which the spindle is connected. Be careful when using the spindle ENB signal in the rigid tapping release sequence.
- Be careful not to issue a spindle mode switching command from one path and a rigid tapping command from another path to the same spindle simultaneously (for example, using an inter-path wait M code).

Example: The Cs contour control function (enabled if bit 2 (SCS) of parameter No.8133 is 1) is executed on the first spindle in path 1. After that, a rigid tapping command is issued from path 2 to that spindle.

Commands in path 1		Commands in path 2
:		:
M20 ; (Enter Cs contour control mode)		:
G00 C90.0 ; (C-axis positioning)		:
M21 ; (Exit from Cs contour control mode)		:
M12000 ;	← Waiting →	M12000 ;
:		M29 S_ P11 ; (Rigid tapping)
:		G84 X_ Z_ F_ ;

M12000: Inter-path wait M code (see descriptions of parameter Nos. 8110 and 8111)
Parameter No.3781 for 1st spindle of path 1 ＝ 11

- If this function is used to carry out rigid tapping on a spindle connected to a path other than a command issuing path, the settings in the path to which the spindle is connected are applied to the following parameters.
  No.5214 (synchronization error width)
  Nos.5221 to 5224 (number of gear teeth on spindle side)
  Nos.5231 to 5234 (number of gear teeth on PC side)
  Nos.5241 to 5244 (maximum rotation speed)
  Nos.5261 to 5264 (acceleration/deceleration constant)
  Nos.5271 to 5274 (acceleration/deceleration constant in extraction)
  Nos.5280 to 5284 (loop gain)
  No.5301, No.5303, No.5305 (spindle effective area (for in-position check))
  No.5311, No.5351, No.5355 (excessive error during spindle movement)
  No.5313, No.5353, No.5357 (excessive error with spindle at stop)
  Nos.5321 to 5324, 5325, 5326, 5327, 5328 (backlash)

10.13.11 Diagnosis Data

For rigid tapping adjustment, the diagnosis screen displays information related to rigid tapping.

- Display of command pulses and position deviation amounts

- Spindle position deviation → Diagnosis data No. 0450
- Number of command pulses distributed to the spindle (momentary value) → Diagnosis data No. 0451
- Cumulative number of command pulses distributed to the spindle → Diagnosis data No. 0454

- Display of rigid tapping synchronization error

- Spindle-converted move command difference → Diagnosis data No. 0455
- Spindle-converted position deviation difference → Diagnosis data No. 0456
- Synchronization error range → Diagnosis data No. 0457
- Spindle-converted move amount difference (maximum value) → Diagnosis data No. 0460
- Spindle-converted machine position difference (momentary value) → Diagnosis data No. 0461
- Spindle-converted machine position difference (maximum value) → Diagnosis data No. 0462

$$\text{Spindle - converted move command difference} = \Sigma \frac{\text{spindle move command}}{\text{gear ratio}} -$$

$$\Sigma \frac{(\text{tapping axis move command}) \times \text{number of pulses per spindle revolution}}{\text{thread lead}}$$

$$\text{Spindle - converted position deviation difference} = \frac{\text{spindle position deviation}}{\text{gear ratio}} -$$

$$\frac{(\text{drilling axis position deviation}) \times \text{number of pulses per spindle revolution}}{\text{thread lead}}$$

$$\text{Synchronization error range} =$$
$$(\text{maximum spindle - converted position deviation difference on the positive side}) -$$
$$(\text{maximum spindle - converted position deviation difference on the negative side})$$

$$\text{Drilling axis machine position} \square \sum [\text{drilling axis move command}] - \text{drilling axis position deviation}$$

$$\text{Spindle machine position} \square \sum [\text{spindle move command}] - \text{spindle position deviation}$$

$$\text{Spindle - converted machine position deviation difference} \square$$
$$\frac{\text{drilling axis machine position} \times \text{number of pulses per spindle revolution}}{\text{thread lead}} \square \frac{\text{spindle machine position}}{\text{gear ratio}}$$

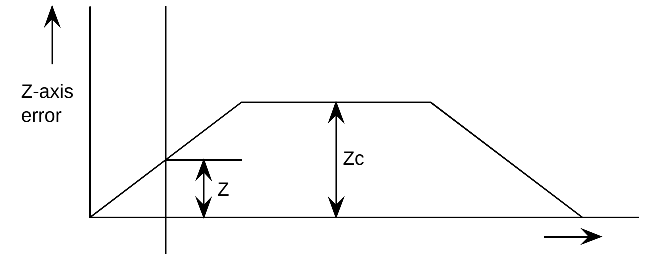
If a maximum allowable synchronization error range is set in parameter No. 5214, the position deviation alarm during spindle movement (alarm SP0741) is issued to indicate that the set synchronization error range has been exceeded. (If 0 is set in parameter No. 5214, no check is performed to detect whether the synchronization error range has been exceeded.)

- Error difference display

- Momentary error difference between the spindle and drilling axis → Diagnosis data No. 0452
- Maximum error difference between the spindle and drilling axis → Diagnosis data No. 0453

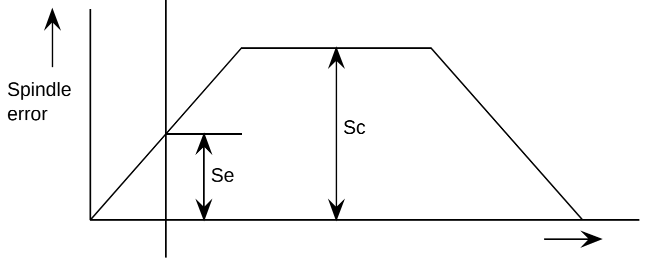
Diagnosis display No. 0452 is cleared to 0 when rigid tapping mode is set or canceled, and diagnosis data No. 0453 is cleared to 0 in the positioning of the rigid tapping cycle.

The following figure shows the tapping axis as the Z axis.



$$Zc = \frac{Speed}{60} \times \frac{1}{Gain} \times \frac{1}{Detection\ unit} \times 10^2 \text{ (Theoretical value)}$$

Ze = Z-axis error counts (measured value)



$$Sc = \frac{Speed \times 360}{60} \times \frac{1}{Gain} \times \frac{1}{Detection\ unit} \times 10^2 \text{ (Theoretical value)}$$

Se = Spindle error counts (measured value)

(Speed : mm/min or inch/min
Gain : 0.01sec^{-1}
Detection unit : mm,inch,or deg
Speed : min^{-1})

$$\Delta Z = \frac{Z}{Zc} \times 100\ [\%]$$

$$\Delta S = \frac{Se}{Sc} \times 100\ [\%]$$

The error difference between the spindle and Z axis can be obtained by ΔS-ΔZ. This value is sampled at intervals of 8 ms.

- Display of spindle position data

Position coder signal pulse data from the spindle one-rotation signal is displayed.

- Position coder signal pulse data of each spindle → Diagnosis data No.0445

NOTE

1 For an unconnected spindle, 0 is displayed.

2 For display of this data, the conditions below must be satisfied.

<1>Serial spindle

<2>Bit 1 (SPP) of parameter No. 3117 must be set to 1.

<3>The state where the serial spindle has detected a one-rotation signal must be set.

To detect a one-rotation signal accurately, spindle orientation must be performed. This orientation operation needs to be performed only once after the power is turned on.

Whether a one-rotation signal has been detected can be known by checking position coder one-rotation signal detection status signal (serial spindle) PC1DTA to PC1DTD<Fn047.0, Fn051.0, Fn170.0, Fn268.0>.

Diagnosis data

- Spindle position data

0445	SPINDLE POSITION DATA

Position coder signal pulse data from the spindle one-rotation signal of each spindle (serial spindle only)

[Unit] Pulse

- Spindle position deviation

0450	SPINDLE MOTION ERROR

Spindle position deviation during rigid tapping

[Unit] Pulse

- Number of pulses distributed to the spindle

0451	SPINDLE MOTION PULSE

Number of pulses distributed to the spindle during rigid tapping

[Unit] Pulse

- Error difference between the spindle and drilling axis (momentary value)

0452	RIGID ERROR

Momentary error difference between the spindle and drilling axis during rigid tapping (signed)

[Unit] %

- Error difference between the spindle and drilling axis (maximum)

0453	RIGID ERROR (MAX)

Maximum error difference between the spindle and drilling axis during rigid tapping (absolute value)

[Unit] %

- Cumulative number of pulses distributed to the spindle during rigid tapping

0454	SPINDLE PULSE (SUM)

Cumulative number of pulses distributed to the spindle during rigid tapping

[Unit] Pulse

- Spindle-converted move command difference during rigid tapping (momentary value)

0455	SYNC. PULSE (SUM)

Momentary spindle-converted move during command difference between the spindle and the drilling axis during rigid tapping

[Unit] Pulse

- Spindle-converted position deviation difference during rigid tapping (momentary value)

0456	SYNC. ERROR

Momentary spindle-converted position deviation difference between the spindle and the drilling axis during rigid tapping

[Unit] Pulse

- Synchronization error range during rigid tapping (maximum value)

0457	SYNC. WIDTH

Synchronization error range during rigid tapping (maximum value)

[Unit] Pulse

- Spindle-converted move command difference during rigid tapping (maximum value)

0460	SYNC. PULSE (MAX)

Maximum spindle-converted move during command difference between the spindle and the drilling axis during rigid tapping

[Unit] Pulse

- Spindle-converted machine position difference during rigid tapping (momentary value)

0461	MACHIN POS ERROR

Spindle-converted machine position difference during rigid tapping (momentary value)

[Unit] Pulse

- Spindle-converted machine position difference during rigid tapping (maximum value)

0462	MACHIN POS ERROR (MAX)

Spindle-converted machine position difference during rigid tapping (maximum value)

[Unit] Pulse

10.13.12 Command Format

T

Command format for the lathe system

The rigid tapping mode can be specified in one of two formats: FANUC Series 16 program format and FANUC Series 15 program format. A usable format can be selected by parameter setting.

Parameter		Executable command format
Bit 1 (FCV) of parameter No. 0001	Bit 3 (F16) of parameter No. 5102	
0	-	FANUC Series 16 program format only (The number of repeats is specified using address K.)
1	0	FANUC Series 15 program format or FANUC Series 16 program format (The number of repeats is specified using address L.)
1	1	FANUC Series 16 program format only (The number of repeats is specified using address L.)

This subsection describes the FANUC Series 16 program command format for rigid tapping on the lathe system. For the machining center system, refer to the description of "RIGID TAPPING" in the Operator's Manual.

- G84 : Face tapping cycle

The first axis of a plane is a drilling axis, and the other axes are positioning axes.

Bit 0 (RTX) of parameter No. 5209	Plane selection	Drilling axis
0	G17: Xp-Yp plane	Xp
	G18: Zp-Xp plane	Zp
	G19: Yp-Zp plane	Yp
1 (Note)		Zp

Xp : X axis or its parallel axis
Yp : Y axis or its parallel axis
Zp : Z axis or its parallel axis
Note) These are invalid in the FANUC Series 15 program format.

1. When the bit 1 (FCV) of parameter No. 0001 is set to 0 :
   G84 X_C_ Z_ R_ P_ F_ K_ M_ ;
   G84 : Face tapping cycle G code
   X C : Tapping position
   Z : Position of the bottom of a hole
   R : Position of an R point (incremental command at all times)
   P : Dwell time at the bottom of a hole
   F : Cutting feedrate
   K : Number of repeats
   M : M code for C-axis clamping

2. When the bit 1 (FCV) of parameter No. 0001 is set to 1 :
   G84 X_C_ Z_ R_ P_ F_ L_ M_ ;
   G84 : Face tapping cycle G code
   X C : Tapping position
   Z : Position of the bottom of a hole
   R : Position of an R point (incremental command at all times)
   P : Dwell time at the bottom of a hole
   F : Cutting feedrate
   L : Number of repeats
   M : M code for C-axis clamping

- G88 : Side tapping cycle

The second axis of a plane is a drilling axis, and the other axes are positioning axes.

Bit 0 (RTX) of parameter No. 5209	Plane selection	Drilling axis
0	G17: Xp-Yp plane	Yp
	G18: Zp-Xp plane	Xp
	G19: Yp-Zp plane	Zp
1 (Note)		Xp

Xp : X axis or its parallel axis
Yp : Y axis or its parallel axis
Zp : Z axis or its parallel axis
Note) These are invalid in the FANUC Series 15 program format.

1. When the bit 1 (FCV) of parameter No. 0001 is set to 0 :
   G88 Z_C_ X_ R_ P_ F_ K_ M_ ;
   G88 : Side tapping cycle G code
   Z C : Tapping position
   X : Position of the bottom of a hole
   R : Position of an R point (incremental command at all times)
   P : Dwell time at the bottom of a hole
   F : Cutting feedrate
   K : Number of repeats
   M : M code for C-axis clamping
2. When the bit 1 (FCV) of parameter No. 0001 is set to 1 :
   G88 Z_C_ X_ R_ P_ F_ L_ M_ ;
   G88 : Side tapping cycle G code
   Z C : Tapping position
   X : Position of the bottom of a hole
   R : Position of an R point (incremental command at all times)
   P : Dwell time at the bottom of a hole
   F : Cutting feedrate
   L : Number of repeats
   M : M code for C-axis clamping

- Method of specification

The rigid tapping mode can be specified using one of three methods:

- Specification of M29S_ before specifying a tapping cycle
- Specification of M29S_ in the same block
- Enabling rigid tapping to be performed without specifying M29S_

When using the third method, specify S_ either before or in a block containing G84 (G88).
Thus, the spindle stops, after which the tapping cycle specified next is placed in rigid tapping mode.
When Spindle selection by address P of Multi spindle is enabled, if S is commanded at G84(G88) block, alarm PS5305, "ILLEGAL SPINDLE NUMBER" is issued. Command S before G84(G88) block.

- Specifying M29 before a block containing G84 (G88)

```
M29    S_;
G84(G88) X_C_(Z_C_) Z_(X_) R_ P_ F_ K_ M_;   ┐
X_C_(Z_C_);                                  │
X_C_(Z_C_);                                  │ Rigid tapping mode
        :                                    │
G80;                                         ┘
```

- Specifying M29 and G84 (G88) in the same block
  (However, the M code for C-axis clamping cannot be specified.)

```
G84(G88) X_C_(Z_C_) Z_(X_) R_ P_ F_ K_ M29 S_;  ┐
X_C_(Z_C_);                                     │
X_C_(Z_C_);                                     │ Rigid tapping mode
        :                                       │
G80;                                            ┘
```

- Converting G84 (G88) to a G code for rigid tapping
  (by setting bit 0 (G84) of parameter No. 5200 to 1)

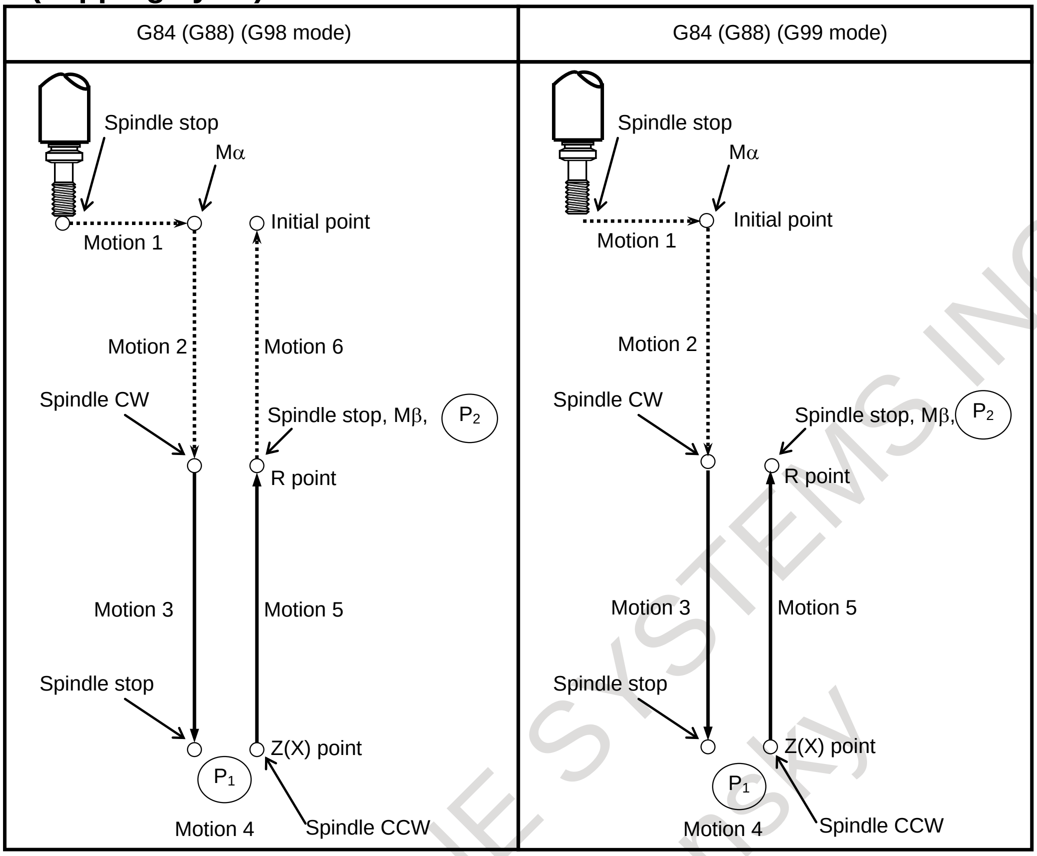
```
G84(G88) X_C_(Z_C_) Z_(X_) R_ P_ F_ K_ S_ M_;   ┐
X_C_(Z_C_);                                     │
X_C_(Z_C_);                                     │ Rigid tapping mode
        :                                       │
G80;                                            ┘
```

- Notes on the lathe system

NOTE

1. In feed per minute mode, F_/S**** determines a thread lead. In feed per revolution mode, F_ specifies a thread lead.
2. S**** must specify a value that does not exceed the value set in the maximum spindle speed parameters Nos. 5241 to 5244 for the gear to be used. Otherwise, an alarm PS0200, "ILLEGAL S CODE COMMAND" is issued in a block containing G84 (G88).
3. F_ must specify a value that does not exceed the maximum cutting feedrate. When 0 is specified, an alarm PS0201, "FEEDRATE NOT FOUND IN RIGID TAP" is issued.
4. Between M29 and G84 (G88), S and a command for movement along an axis must not be specified. Further, M29 must not be specified in a tapping cycle. Otherwise, alarms PS0203, "PROGRAM MISS AT RIGID TAPPING" and PS0204, "ILLEGAL AXIS OPERATION" are issued, respectively.
5. In the rigid tapping mode, G84 and G88 must not be specified in succession. Cancel the rigid tapping mode before specifying G84 and G88 in succession.

G84, G88 (Tapping cycle)



Rapid traverse
Cutting feed
P1 Dwell specified by address P in the program
P2 Dwell set by parameter No. 5111 (not performed in the FANUC Series 15 program format)
Mα Output of an M code for C-axis clamping
Mβ Output of an M code for C-axis unclamping (β=α+1) (not performed in the FANUC Series 15 program format)
Supplement: P1, Mα, Mβ, and P2, when not specified, are not executed and output.

> ⚠ **CAUTION**
> For cutting feed along the Z(X)-axis and override in extraction, see Subsection "Override".

> **NOTE**
> G code system A does not include G98 (return to initial level) and G99 (return to R point level). Return to the initial level is always used.

10.13.13 Position Control Loop Gain Parameter Switching

In rigid tapping, the position control loop gain of the drilling axis must match that of the spindle. Specifically, change the position control loop gain of the drilling axis at the following timings:

(1) A change from a position control loop gain for normal operation to a position control loop gain for rigid tapping must be made when the cutting along the drilling axis is started (when motion 3 is started).

(2) A change from a position control loop gain for rigid tapping to a position control loop gain for normal operation must be made when the operation after the completion of extraction along the drilling axis is started (when motion 6 is started).

While a position control loop gain for rigid tapping is selected, the parameters dedicated to rigid tapping are used for the following parameters for the drilling axis:

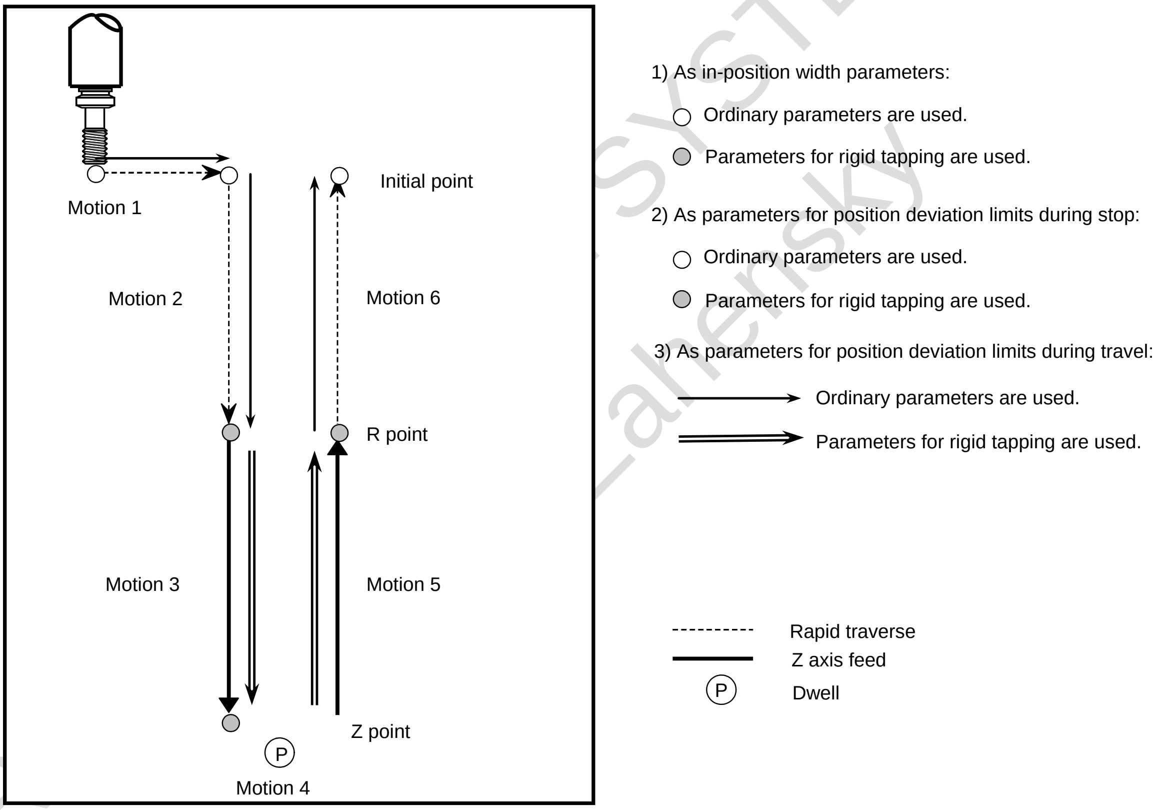
(1) In-position width
(Parameters Nos.5300, 5302, 5304, and 5306)

(2) Position deviation limit during stop
(Parameters Nos.5312, 5352, 5356, and 5360)

(3) Position deviation limit during travel
(Parameters Nos.5310, 5350, 5354, and 5358)

The above parameters dedicated to rigid tapping are usually used to specify the in-position width at the R point for motion 2. However, an ordinary parameter No. 1826 can be selected by using bit 4 (IRR) of parameter No. 5202.

> **NOTE**
> If bit 5 (NCI) of parameter No. 1601 is set to 1, no in-position check is made at the R point for motion 2 except when bit 1 (RIP) of parameter No. 5209 is set to 1.



10.13.14 Signal

10.13.14.1 Signals for the rigid tapping function

Rigid tapping signal RGTAP<Gn061.0>

[Classification] Input signal

[Function] When M29 (miscellaneous function for preparation for rigid tapping) is specified, the PMC enters rigid tapping mode, then turns on this signal to notify the CNC.

1 : The PMC enters in rigid tapping mode.

0 : The PMC does not enter rigid tapping mode.

For an explanation of placing the PMC in rigid tapping mode, see the description of the interface with the PMC, given later.

This signal posts whether the PMC has entered rigid tapping mode. If this signal is not set to 1, even when M29 is specified, alarm (PS0205) "RIGID MODE DI SIGNAL OFF" is issued in a G84/G74(machining center system) or G84/G88(lathe system) block.

Spindle rotation direction signals RGSPM,RGSPP<Fn065.1,0>

[Classification] Output signal

[Function] During rigid tapping, these signals notify the PMC of whether the spindle is rotating in the forward or reverse direction.
→ During rigid tapping, the spindle is:

RGSPP 1 : Rotating in the forward direction (CW).
0 : Not rotating in the forward direction (CW).

RGSPM 1 : Rotating in the reverse direction (CCW).
0 : Not rotating in the reverse direction (CCW).

[Output cond.] These signals are output when the spindle is rotating in rigid tapping mode. This means that, even in rigid tapping mode, these signals are not output, for example, when the spindle is being positioned to a hole position, or a dwell operation is in progress at the bottom of a hole or at an R point.
These signals are not output in the feed hold state or single block stop state. When the spindle is placed in the interlock stop state, machine lock state, or Z-axis ignore state, however, the spindle is not regarded as having stopped; these signals are output.
These signals are valid only in rigid tapping mode. In normal spindle control, these signals are not output; both RGSPP and RGSPM are set to 0.

Rigid tapping in-progress signal RTAP<Fn076.3>

[Classification] Output signal

[Function] This signal notifies the PMC that rigid tapping mode is active.

RTAP 1 : Rigid tapping mode is currently active.
0 : Rigid tapping mode is not currently active.
By latching M29, the PMC knows that rigid tapping mode has been specified, and thus performs the required processing on the PMC side. This signal can substitute for the latching of M29. Even in this case, however, FIN for M29 cannot be omitted.

10.13.14.2 Signals related to S code output

Spindle enable signal ENB<Fn001.4>

[Classification] Output signal

[Function] This signal post whether the spindle output is 0. In rigid tapping mode, this signal is used to cancel rigid tapping in a PMC sequence associated with rigid tapping. For details, see the explanation of the interface with the PMC, given later.

Spindle-speed function code signals (binary output) S00 to S31<Fn022 to Fn025>
Spindle-speed function strobe signals SF<Fn007.2>

[Classification] Output signal

[Function] These signals send S codes specified for the CNC, in binary format, to the PMC.

[Output cond.] When an S code is specified, the specified value is output, in binary format, with these signals. When the new spindle speed data is fully set, the SF signal is set to 1.
Before rigid tapping can be performed, however, parameter setting is required to output these signals, as described below.

Machining center system :
SF output depends on the gear selection method, as described below.
[1] M-type gear selection method
SF output depends on bit 6 (SFA) of parameter No. 3705.

[2] T-type gear selection method
SF output depends on the setting of bit 5 (NSF) of parameter No. 3705.

Lathe system :
The following parameter needs to be set to output S codes and SF:
Bit 4 (EVS) of parameter No. 3705 is set to 1

In rigid tapping, when SF is to be used by the PMC to read an S code output signal for gear switching or output switching, set the above parameters as required.

NOTE

1 The timing charts, given later, give examples of gear switching by setting the parameters as follows:
Machining center system : SFA=0,NSF=0
Lathe system : EVS=1

2 When the constant surface speed control function is being used, an S code (specifying a surface speed) used for constant surface control (G96) is output. Such an S code can be distinguished from an S code used for specifying a rotation speed. One method is to use, for example, the constant surface speed control in-progress signal <F002.2> for the processing performed on the PMC side. Another method is to mask the S code and SF signal, output by setting bit 0 (ESF) of parameter No. 3705.

10.13.14.3 Signals related to gear switching

Gear selection signals (output) GR3O,GR2O,GR1O<Fn034.2,1,0>

M

[Classification] Output signal

[Operation] When M-type gear selection is being used, these signals are used in a PMC sequence for rigid tapping.
The signals post, to the PMC, information about a spindle gear to be used, according to the value of S_ specified at the execution of G84 (G74).
When gear switching becomes necessary, the states of the signals change together with the SF signal.
The PMC should perform gear switching according to the information posted by the gear selection signals.
Reference information:
The table below indicates the relationship between the output signals and gear selection.

	GR3O	GR2O	GR1O
1st (low) speed gear	0	0	1
2nd (medium) speed gear	0	1	0
3rd (high) speed gear	1	0	0

Gear selection signals (input) GR2,GR1<Gn028.2,1>

[Classification] Input signal

[Operation] When T-type gear selection is being used, these signals are used in a PMC sequence for rigid tapping.
The signals post, to the CNC, information about a spindle gear to be used.
Reference information:
The table below shows the relationship between the output signals and spindle gear selection.

	GR1	GR2
1st (low) speed gear	0	0
2nd (medium) speed gear	1	0
3rd (high) speed gear	0	1
4th (high) speed gear	1	1

The serial spindle clutch/gear selection signals <G070.3,2> must be set in addition to the setting of the gear selection signal described above.

NOTE
In machining center system rigid tapping, the specification of the 4th (high) speed gear is invalid. If specified, the system assumes that the 3rd (high) speed gear has been specified.

10.13.14.4 Signals related to the addition of multi-spindle control

Spindle enable signals ENB<Fn001.4>,ENB2,ENB3<Fn038.2,3>,ENB4<Fn039.1>

[Classification] Output signal

[Function] These signals post whether the spindle output to each spindle is 0 in multi-spindle control.
In rigid tapping mode, these signals are used to cancel rigid tapping in a PMC sequence associated with rigid tapping.
For details, see the explanation of the interface with the PMC, given later.

Gear selection signals (input)
GR2,GR1<Gn028.2,1>,GR22,GR21<Gn029.1,0>,GR32,GR31<Gn029.3,2>, GR42,GR41<Gn031.5,4>

[Classification] Input signal

[Operation] When the T-type gear selection method is employed in multi-spindle control, these signals are used with a PMC sequence for rigid tapping.
The signals post, to the CNC, information about a spindle gear to be used.
Reference information:
The table below shows the relationship between the output signals and spindle gear selection.

	GRs1	GRs2
1st (low) speed gear	0	0
2nd (medium) speed gear	1	0
3rd (high) speed gear	0	1
4th (high) speed gear	1	1

(s is -, 2, 3, or 4.)

The serial spindle clutch/gear selection signals (G070.3,2 for the first spindle, G074.3,2 for the second spindle, G204.3,2 for the third spindle, and G266.3,2 for the fourth spindle) must be set in addition to the setting of the gear selection signal described above.

NOTE
In machining center system rigid tapping, the specification of the 4th (high) speed gear is invalid. If specified, the system assumes that the 3rd (high) speed gear has been specified.

Spindle selection signals
SWS1<Gn027.0>,SWS2<Gn027.1>,SWS3<Gn027.2>,SWS4<Gn026.3>
Rigid tapping spindle selection signals
RGTSP1<Gn061.4>,RGTSP2<Gn061.5>,RGTSP3<Gn061.6>,RGTSP4<Gn061.7>

T

[Classification] Input signal

[Operation] SWS1 to SWS4 are used to transfer spindle commands when the multi-spindle control is enabled (bit 3 (MSP) of parameter No.8133 is 1). In rigid tapping, the signals can be shared to select a spindle to be used for rigid tapping. (The signals can be used for this purpose when bit 7 (SRS) of parameter No. 5200 is set to 0.)
RGTSP1, RGTSP2, RGTSP3, and RGTSP4 are used to select a spindle used for rigid tapping, independently of the SWS1 to SWS4 signals, when the multi-spindle control is enabled (bit 3 (MSP) of parameter No.8133 is 1). (The signals can be used when bit 7 (SRS) of parameter No. 5200 is set to 1. These signals are supported only by the lathe system.)
See the tables below for details of the settings of these signals.
When bit 7 (SRS) of parameter No. 5200 is set to 0, to select a spindle to be used for rigid tapping, set the signals as indicated below.

Spindle used for rigid tapping	Signal state			
	SWS1	SWS2	SWS3	SWS4
First spindle	"1"	"0"	"0"	"0"
Second spindle	"0"	"1"	"0"	"0"
Third spindle	"0"	"0"	"1"	"0"
Fourth spindle	"0"	"0"	"0"	"1"
Alarm PS0205 is issued.	"0"	"0"	"0"	"0"

When bit 7 (SRS) of parameter No. 5200 is set to 1, to select a spindle to be used for rigid tapping, set the signals as indicated below.

Spindle used for rigid tapping	Signal state			
	RGTSP1	RGTSP2	RGTSP3	RGTSP4
First spindle	"1"	"0"	"0"	"0"
Second spindle	"0"	"1"	"0"	"0"
Third spindle	"0"	"0"	"1"	"0"
Fourth spindle	"0"	"0"	"0"	"1"
Alarm PS0205 is issued.	"0"	"0"	"0"	"0"

NOTE
1 These signals must be applied before the command for rigid tapping (M29 S....; G84 X...) is specified. The states of these signals must not be changed before rigid tapping has been completed.
2 When multiple signals from SWS1 to SWS4 are set to "1" simultaneously, the signals are checked in the order from SWS1 to SWS2 to SWS3 to SWS4, and the spindle corresponding to the signal that is first found to be set to "1" is assumed to be specified.

> **NOTE**
> 3 Similarly, when multiple signals from RGTSP1 to RGTSP4 are set to "1" simultaneously, the signals are checked in the order from RGTSP1 to RGTSP2 to RGTSP3 to RGTSP4, and the spindle corresponding to the signal that is first found to be set to "1" is assumed to be specified.

Spindle-by-spindle stop signals*SSTP1<Gn027.3>,*SSTP2<Gn027.4>,*SSTP3<Gn027.5>, *SSTP4<Gn026.6>

[Classification] Input signal

[Operation] These signals are used to stop each spindle when the multi-spindle control option is enabled (bit 3 (MSP) of parameter No.8133 is 1). In a PMC sequence for rigid tapping, the ENB1, ENB2, ENB3 and ENB4 signals are used. Accordingly, the logic of the signals used for a spindle selected to perform rigid tapping must match the logic of the spindle stop signal *SSTPs.

*SSTP1 1 : The output to the first spindle is not forced to 0 min^{-1}.
0 : 0 min^{-1} is commanded to first spindle.

*SSTP2 1 : The output to the second spindle is not forced to 0 min^{-1}.
0 : 0 min^{-1} is commanded to second spindle.

*SSTP3 1 : The output to the third spindle is not forced to 0 min^{-1}.
0 : 0 min^{-1} is commanded to third spindle.

*SSTP4 1 : The output to the fourth spindle is not forced to 0 min^{-1}.
0 : 0 min^{-1} is commanded to fourth spindle.

Position coder selection signal PC2SLC<Gn028.7>,PC3SLC<Gn026.0>,PC4SLC<Gn026.1>

[Classification] Input signal

[Operation] This signal is used to select the position coder when the multi-spindle control option is enabled (bit 3 (MSP) of parameter No.8133 is 1). Note, however, that it cannot be used with a spindle selected to perform rigid tapping.
For rigid tapping, this signal is not used. Instead, a position loop is constructed by combining the first spindle with the first position coder, by combining the second spindle with the second position coder, by combining the third spindle with the third position coder, or by combining the fourth spindle with the fourth position coder.
However, the display of the actual speed is switched by this signal, even during rigid tapping.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn026		*SSTP4			SWS4		PC4SLC	PC3SLC
Gn027			*SSPT3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
Gn028	PC2SLC					GR2	GR1	
Gn029		*SSTP			GR32	GR31	GR22	GR21
Gn031			GR42	GR41				
Gn061	RGTSP4	RGTSP3	RGTSP2	RGTSP1				RGTAP

	#7	#6	#5	#4	#3	#2	#1	#0
Fn001				ENB				
Fn007						SF		
Fn034						GR3O	GR2O	GR1O
Fn022	S07	S06	S05	S04	S03	S02	S01	S00
Fn023	S15	S14	S13	S12	S11	S10	S09	S08
Fn024	S23	S22	S21	S20	S19	S18	S17	S16
Fn025	S31	S30	S29	S28	S27	S26	S25	S24
Fn038					ENB3	ENB2		
Fn039							ENB4	
Fn065							RGSPM	RGSPP
Fn076					RTAP			

10.13.14.5 Notes on interface with the PMC

The following describes some notes in designing the interface with the PMC. Refer to the following timing chart about ON/OFF of signals.

- Rigid tapping mode management and ENB (ENB2,ENB3,ENB4)

The PMC must manage rigid tapping mode as follows: rigid tapping mode is set using M29, and is canceled upon the issue of a reset or at the falling edge of the spindle enable signal ENB in rigid tapping mode. ENB is used during rigid tapping in this way, so the spindle stop signal *SSTP must not be set to "0".
However, the spindle orientation function (*SSTP and SOR) may be used for gear switching. To do so, ensure that the PMC does not cancel rigid tapping mode on a falling edge of ENB while *SSTP is "0".
Rigid tapping mode may be set on a rising edge of the RTAP signal instead of by using M29, and canceled on a falling edge of the RTAP signal instead of the ENB signal. In rigid tapping under multi -spindle control, use the spindle enable signal ENBs for the spindle engaged in rigid tapping to exercise control.

- Controlling spindle output by the PMC

When the SIND signal is set to "1", spindle output is controlled by the signals (SSIN, SGN, R1I to R12I) output from the PMC.
At this time, the effect of ENB is as described above. In addition, when rigid tapping mode is canceled in a block containing G80;, the momentary rotation of the spindle, caused by a delay in the PMC processing, can result. Accordingly, the PMC's control over spindle output must be disabled in rigid tapping mode by setting SIND to "0".
In rigid tapping under multi-spindle control, ensure, for a similar reason, that in the rigid tapping mode, PMC-based control is disabled (SINDs = "0") for the spindle used for rigid tapping.

- T-type gear selection method

When T-type gear selection is used, the PMC must determine whether gear switching is to be performed, and subsequently perform gear switching as required. For this purpose, each time a spindle-speed function code is specified, the spindle-speed function code read signal (SF) and spindle-speed function code signals (S00 to S31) must be output to the PMC. The required parameter settings are described below.

- Machining center system :
  Set bit 5 (NSF) of parameter No. 3705 to 0 to output SF.
- Lathe system :
  Set bit 4 (EVS) of parameter No. 3705 to 1 to output SF.

- Gear switching timing

In general, a block containing M29 (miscellaneous function for preparation for rigid tapping) specifies S_, S_ being output when a block containing G84/G74 (machining center system) or G84/G88 (lathe system) is executed. This means that gear switching is performed in the block specifying G84 (G74).

- When rigid tapping mode is specified

M29 (miscellaneous function for preparation for rigid tapping) and S_ specify rigid tapping mode. When M29 is accepted by the PMC, the following processing must be performed:

- Stop the spindle when it is rotating.
- Check speed zero signal SSTs if the spindle has been stopped, then set "1" to rigid tapping signal RGTAP <Gn061.0>.
- Activate the spindle motor. Activate the motor so that a positive speed command rotates the spindle in the forward direction (CCW when viewed from the - side of the tapping axis).
- Return FIN at least 250 ms after activation.

> **NOTE**
> The condition "at least 250 ms after activation" results from there being no way of checking the completion of spindle motor activation. Therefore, this wait period serves as an alternative. The time required for activation to be completed varies with the spindle motor and amplifier. Therefore, this value of 250 ms is given as a guideline only.

In an M29 block, S_ is not executed, merely being read in. Spindle output is equivalent to the specification of S0. S_ is executed in a G84/G74 (machining center system) or G84/G88 (lathe system block). The timing chart is shown in the chart indicating the execution of G84/G74 or G84/G88.

- Execution of G84/G74 (machining center system) or G84/G88 (lathe system block)

When M29S_; is specified, S_ is read in, spindle output being equivalent to the specification of S0;. S_ is output when G84/G74 (machining center system) or G84/G88 (lathe system block) is executed. Thus, the processing described below is performed.

- When M-type gear selection is used

When using a machine that features multiple gear stages for use with the spindle motor and spindle, and the newly programmed S_ is outside the previously selected gear range, the spindle-speed function strobe signal SF <F007.2> and gear selection signals (output) GR3O, GR2O, GR1O <F034.2, 1, 0> are output to the PMC.
At this time, perform gear switching at the PMC.

- When T-type gear selection is used

The spindle-speed function strobe signal SF <F007.2> and spindle-speed function code signals S00 to S31 <F022 to F025> are output to the PMC. (However, parameter setting is required to enable output of the S codes and SF signal. See the description of each bit of parameter No. 3705.)
At this time, the PMC must determine whether gear switching is to be performed, and perform gear switching as required. The selected gear must be reflected in the gear selection signals (input) GR2 and GR1 <G028.2, 1> for notification to the CNC.
From GR2 and GR1, the CNC determines which gear is selected.
However, note the difference between the machining center system and lathe system, as described below.

- Machining center system :

Up to three gear stages are supported. If the fourth gear stage is selected, it is assumed that the third gear stage is selected.

- Lathe system :
  Up to four gear stages are supported.
  (A gear selection is posted to the CNC with the gear selection signal for each spindle.)

- C-axis clamp/unclamp

T

When an M code for C-axis clamping is specified in a block specifying G84/G88, the M codes for C-axis clamping/unclamping are output at the following timings:

- The M code for clamping is output at the time of rapid traverse to the R point level (motion 2). When FIN is returned, tapping (motion 3) starts.
- After extraction from the hole bottom to the R point level (motion 5), the M code for unclamping is output. When FIN is returned, dwell or rapid traverse to the initial level (motion 6) starts.

The rigid tapping mode can be specified in three different methods. One method specifies M29 before G84. The second method specifies M29 and G84 in the same block. The third method uses G84 as a rigid tapping G code. In any method, however, PMC processing is the same. (In any case, the M29 code is output.)

10.13.15 Timing Charts for Rigid Tapping Specification

The timing chart for rigid tapping specification depends on the method used to specify rigid tapping mode, the gear selection method (M-type or T-type), and whether to perform gear switching.
From the table, find the appropriate timing chart and apply the information it contains as necessary.

M

Gear selection method M-type T-type	Gear switching	Specification method		
		M29 is specified before G84/G74.	M29 and G84/G74 are specified in the same block.	By parameter setting, G84/G74 is specified as a G code for rigid tapping.
M-type	Not performed	Fig. 10.13.15.1 (a)	Fig. 10.13.15.2 (a)	Fig. 10.13.15.3 (a)
	Performed	Fig. 10.13.15.1 (b)	Fig. 10.13.15.2 (b)	Fig. 10.13.15.3 (b)
T-type	Not performed	Fig. 10.13.15.1 (c)	Fig. 10.13.15.2 (c)	Fig. 10.13.15.3 (c)
	Performed	Fig. 10.13.15.1 (d)	Fig. 10.13.15.2 (d)	Fig. 10.13.15.3 (d)

T

Gear switching	Specification method		
	M29 is specified before G84/G88.	M29 and G84/G88 are specified in the same block.	By parameter setting, G84/G88 is specified as a G code for rigid tapping.
Not performed	Fig. 10.13.15.4 (a)	Fig. 10.13.15.5 (a)	Fig. 10.13.15.6 (a)
Performed	Fig. 10.13.15.4 (b)	Fig. 10.13.15.5 (b)	Fig. 10.13.15.6 (b)

NOTE

For more information about the M/T type gear selection method, see Section "SPINDLE SPEED CONTROL." Note the following:

Machining center system :

When constant surface speed control is not being used and bit 4 (GTT) of parameter No. 3706 is set to 0. → M-type

When constant surface speed control is being used, or bit 4 (GTT) of parameter No. 3706 is set to 1. → T-type

Lathe system : T-type only

10.13.15.1 When M29 is specified before G84/G74

- M type gear selection method

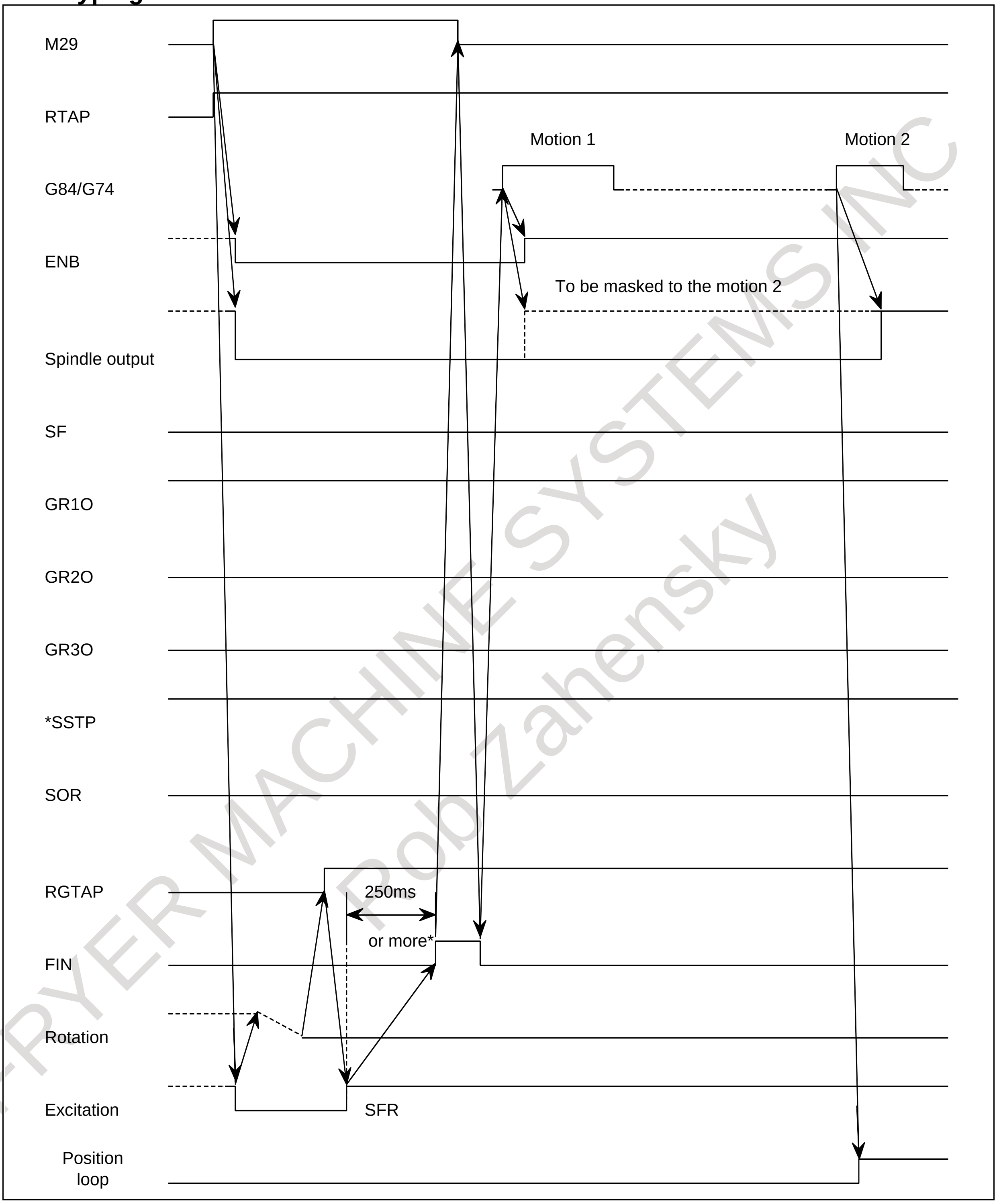


Fig. 10.13.15.1 (a) Gear is not changed

> **NOTE**
> The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

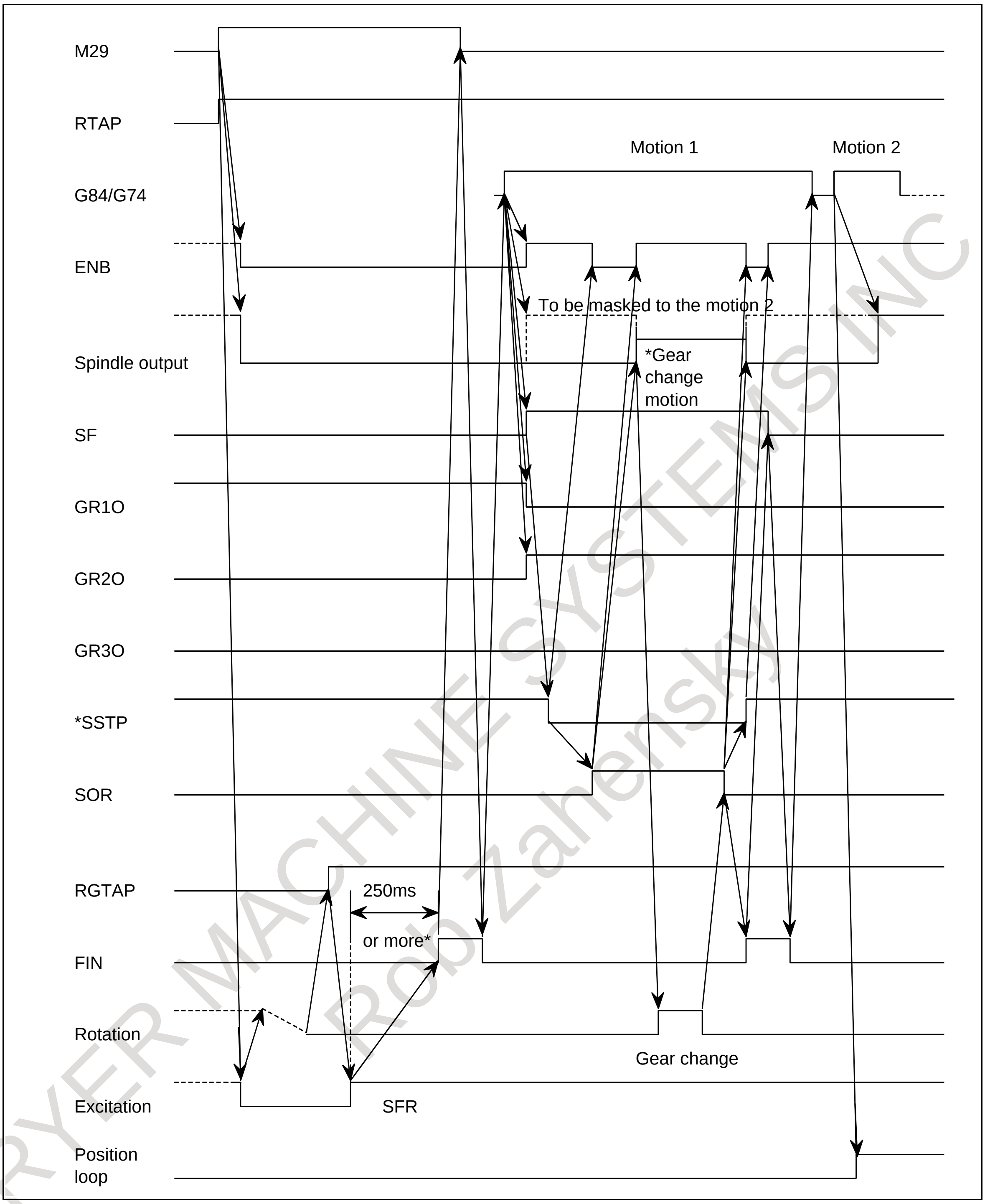


Fig. 10.13.15.1 (b) When gear change is performed (from low to middle gear)

NOTE

1. This time chart show an example where the gear has shifted from low to middle gear. One of the gear select signals (GR1O, GR2O, GR3O) has turned from "1" to "0", and one of the two remaining signals has turned from "0" to "1". This changes the gear.
2. The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

- T type gear selection method

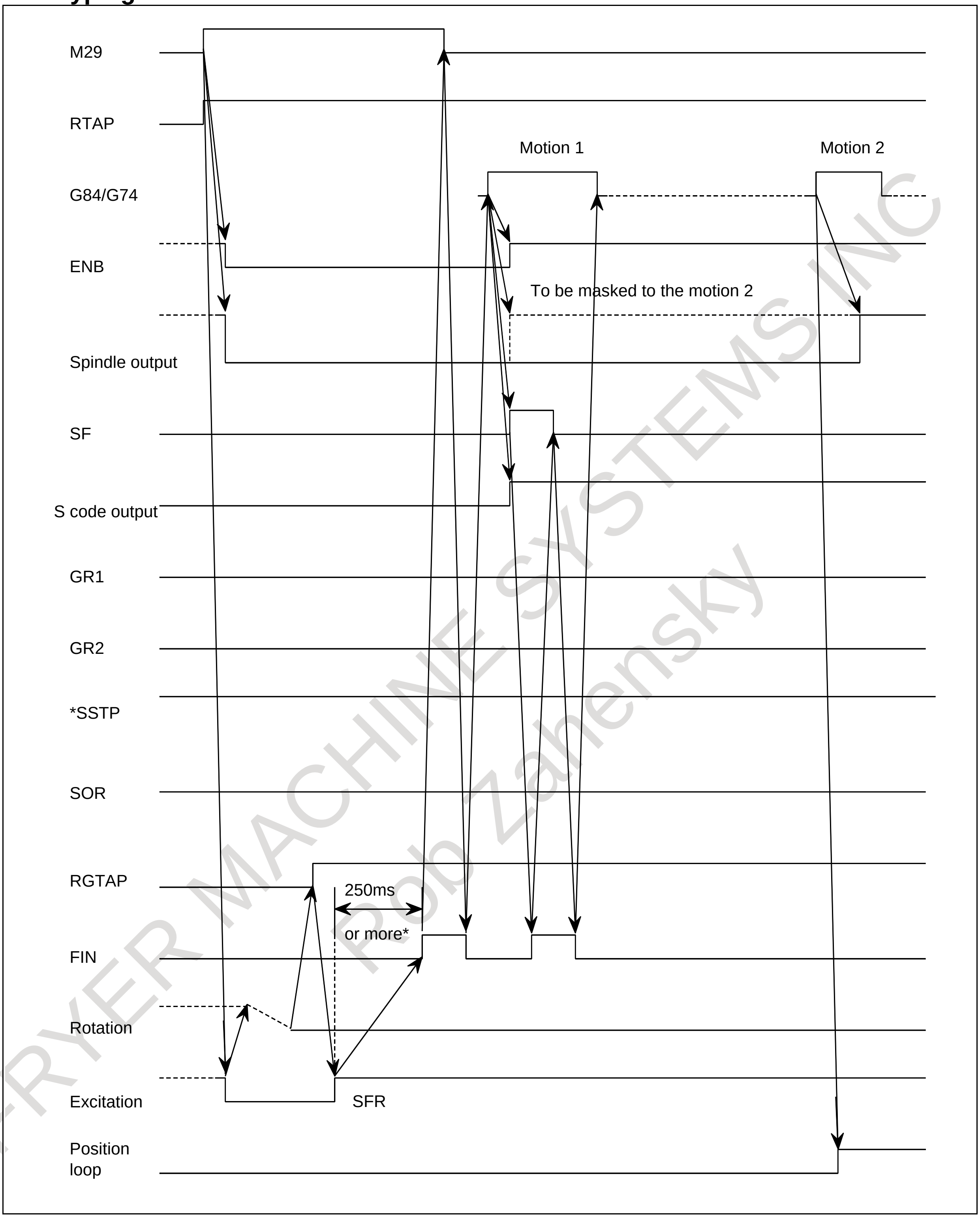


Fig. 10.13.15.1 (c) Gear change is not performed

> **NOTE**
> The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

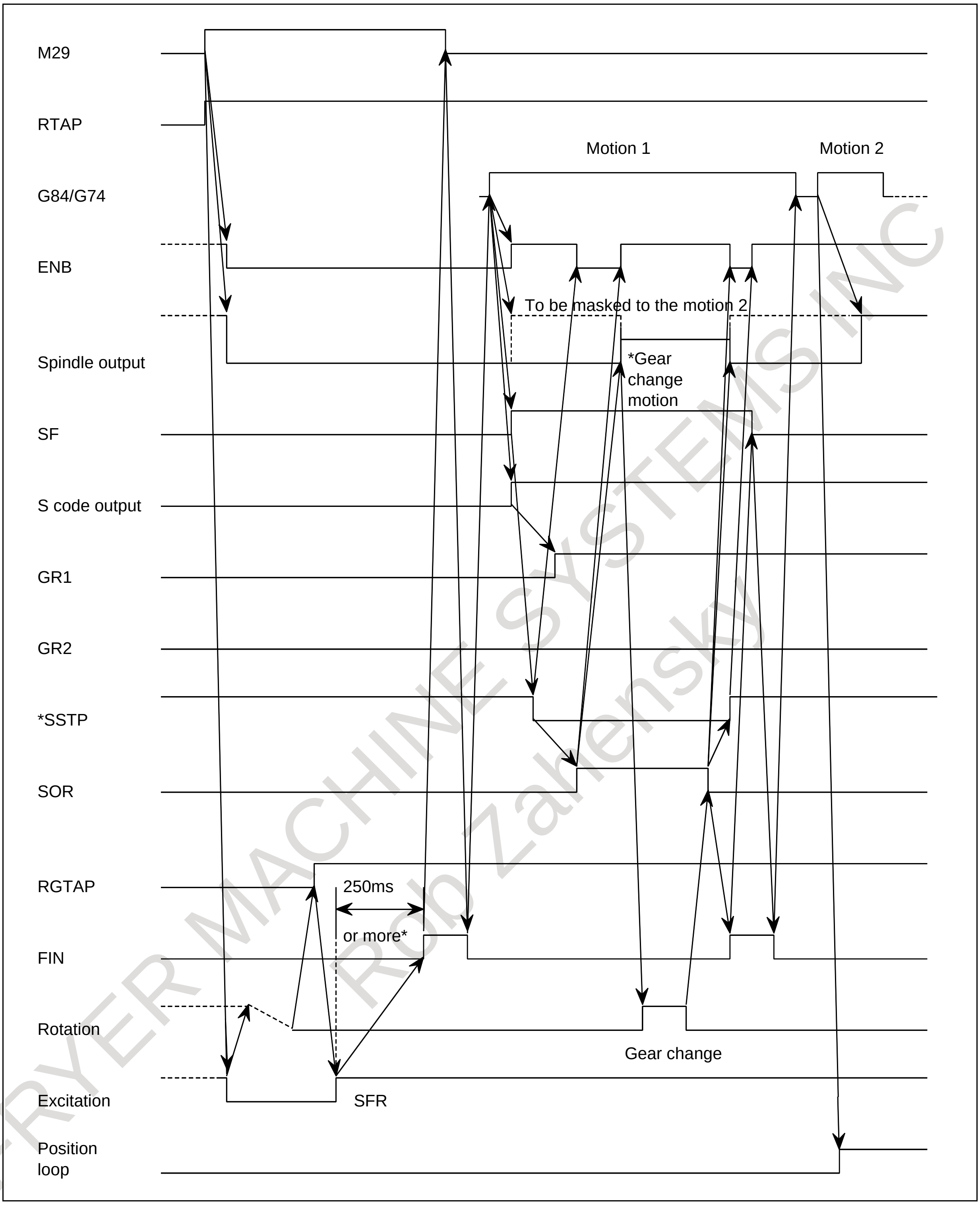


Fig. 10.13.15.1 (d) When gear-change is performed (low to middle gear)

NOTE

1 This time chart shows an example where the gear has changed from low to middle gear. The PMC selects a required gear through an S code output and selects the gear using the GR1 and GR2 signals to inform CNC of the selected gear.

2 The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

10.13.15.2 M29 and G84/G74 are specified in the same block

- **M type gear selection**

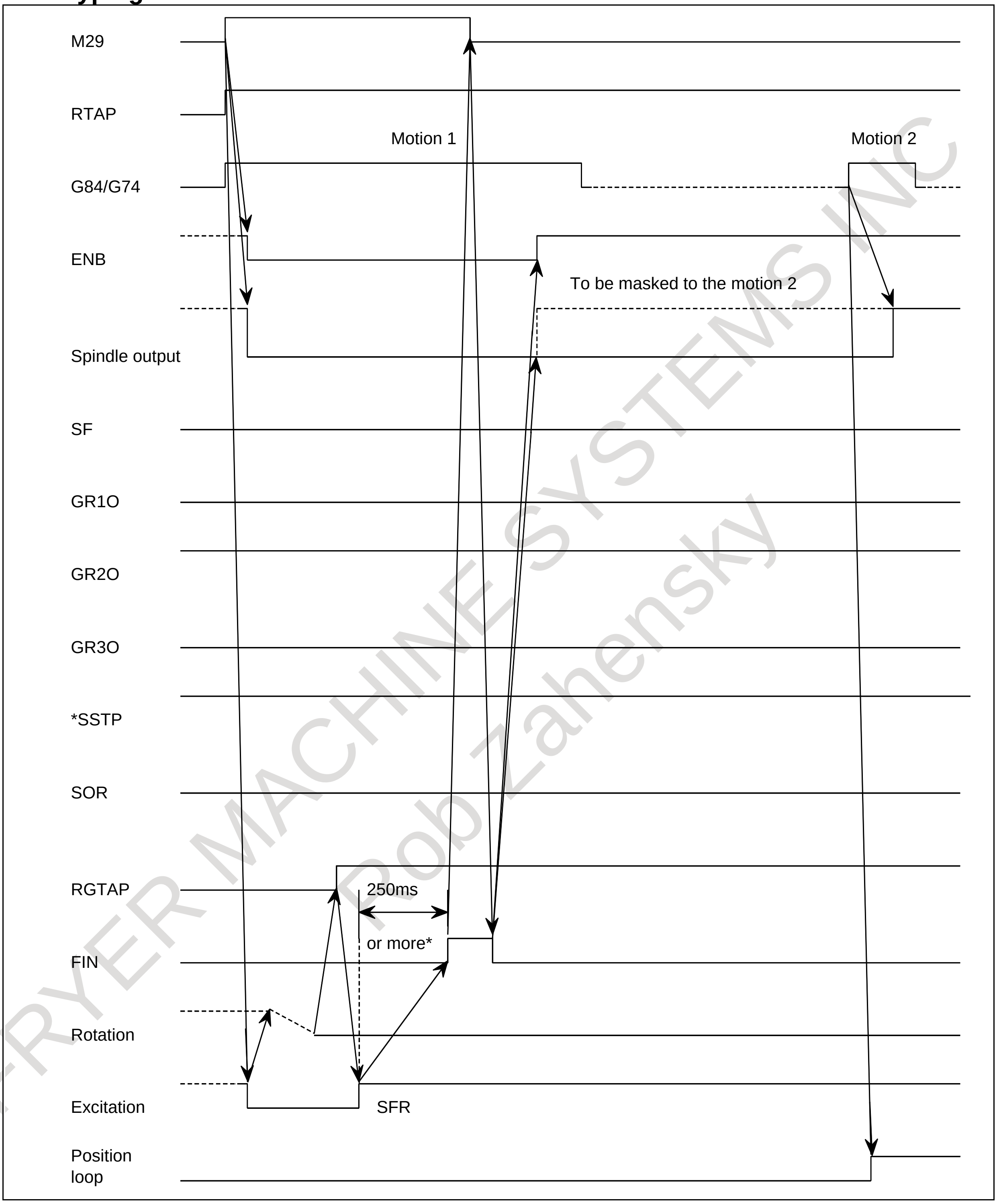


Fig. 10.13.15.2 (a) When gear-change is not performed

> **NOTE**
> The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

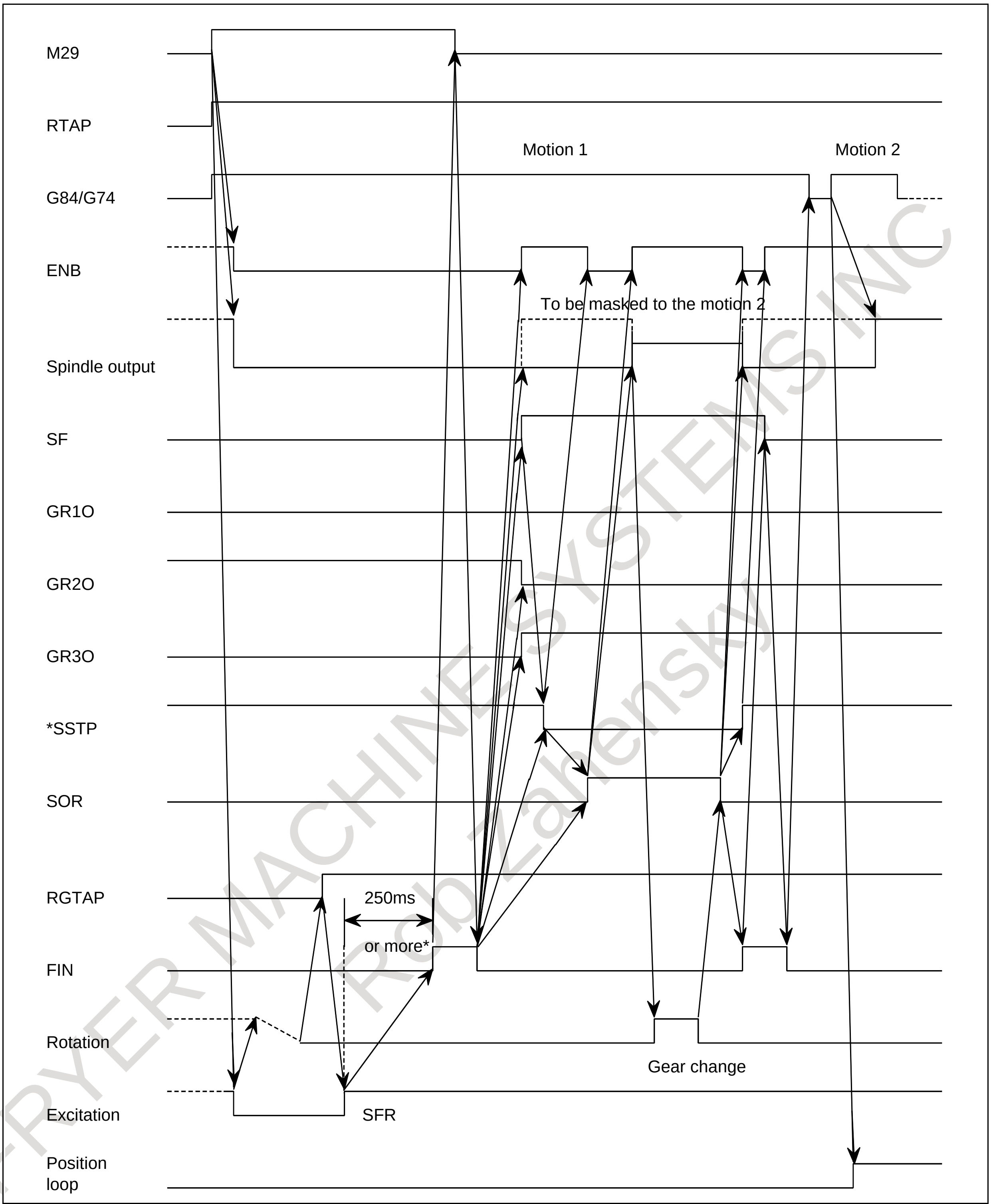


Fig. 10.13.15.2 (b) When gear-change is performed (middle to high gear)

NOTE

1. This time chart shows an example where the gear has shifted from middle to high gear. One of the gear select signals (GR1O, GR2O, GR3O) has turned from "1" to "0", and one of the two remaining signals has tuned from "0" to "1". This changes the gear.
2. The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

- T type gear selection method

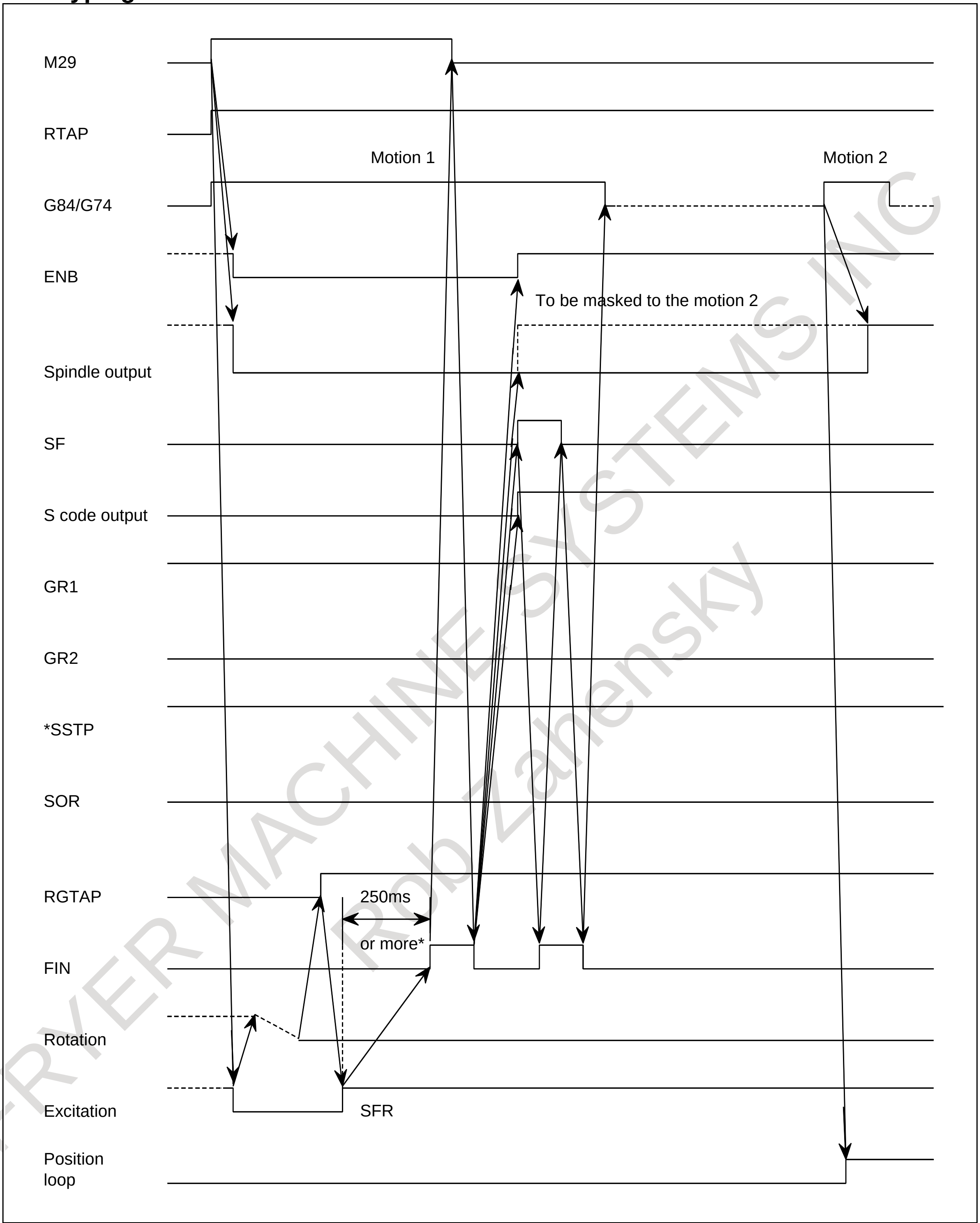


Fig. 10.13.15.2 (c) When gear change is not performed

> **NOTE**
> The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

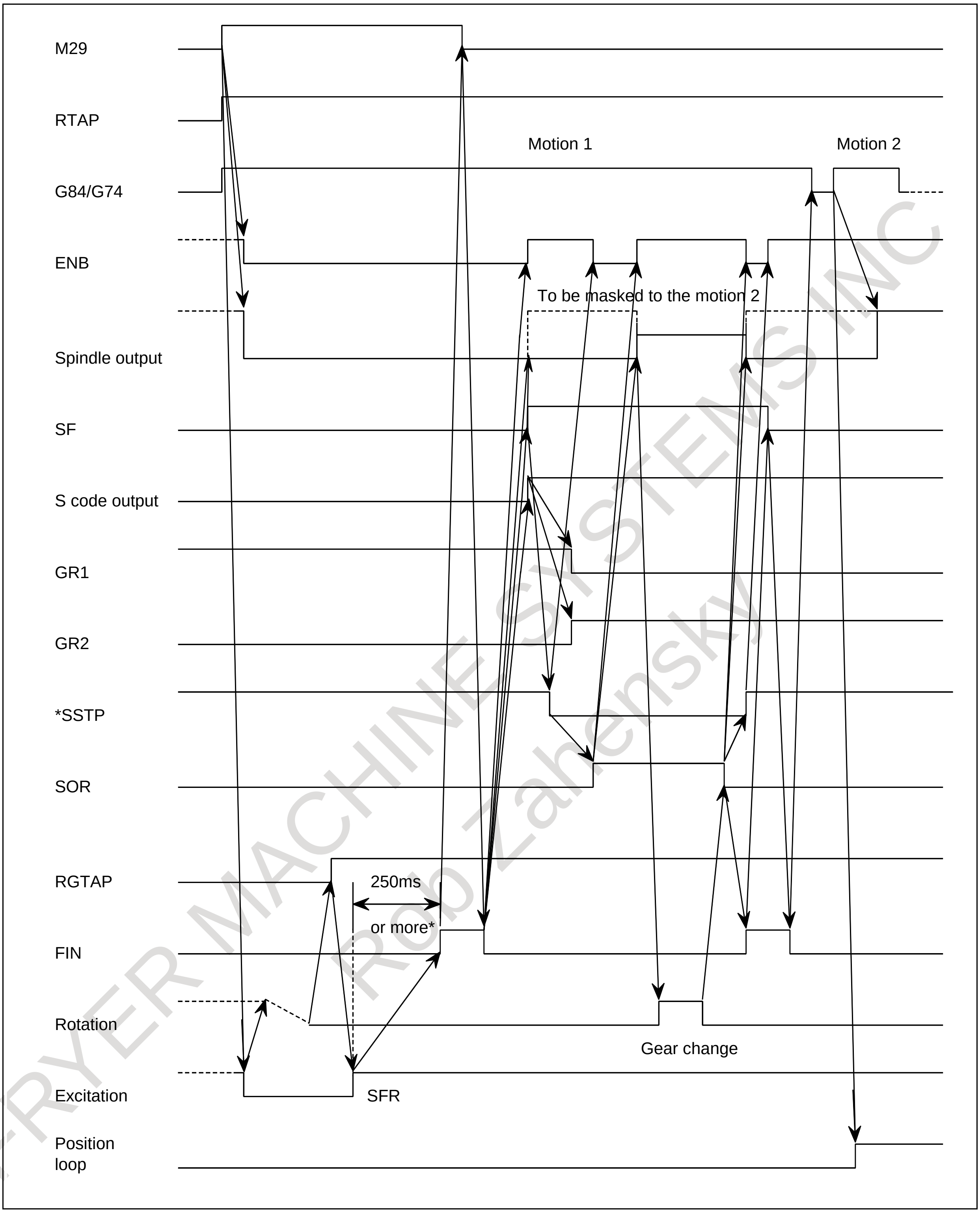


Fig. 10.13.15.2 (d) When gear-change is performed (middle to high gear)

NOTE
1. This time chart shows an example where the gear has changed from middle to high gear. The PMC selects a required gear through an S code output and selects the gear using the GR1 and GR2 signals to inform CNC of the selected gear.
2. The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

10.13.15.3 Specifying G84/G74 for rigid tapping by parameters

- M type gear selection

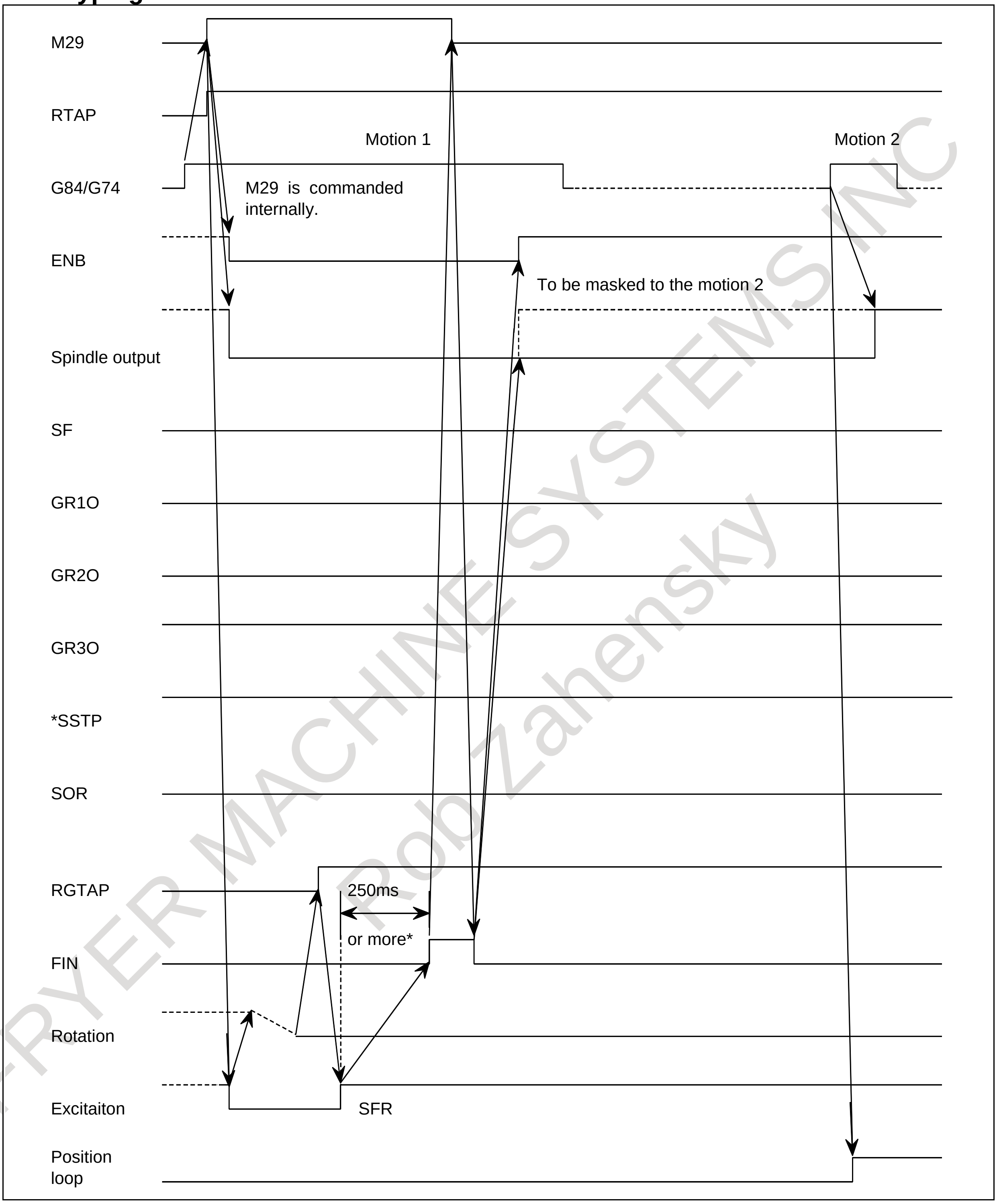


Fig. 10.13.15.3 (a) When gear-change is not performed

> **NOTE**
> The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

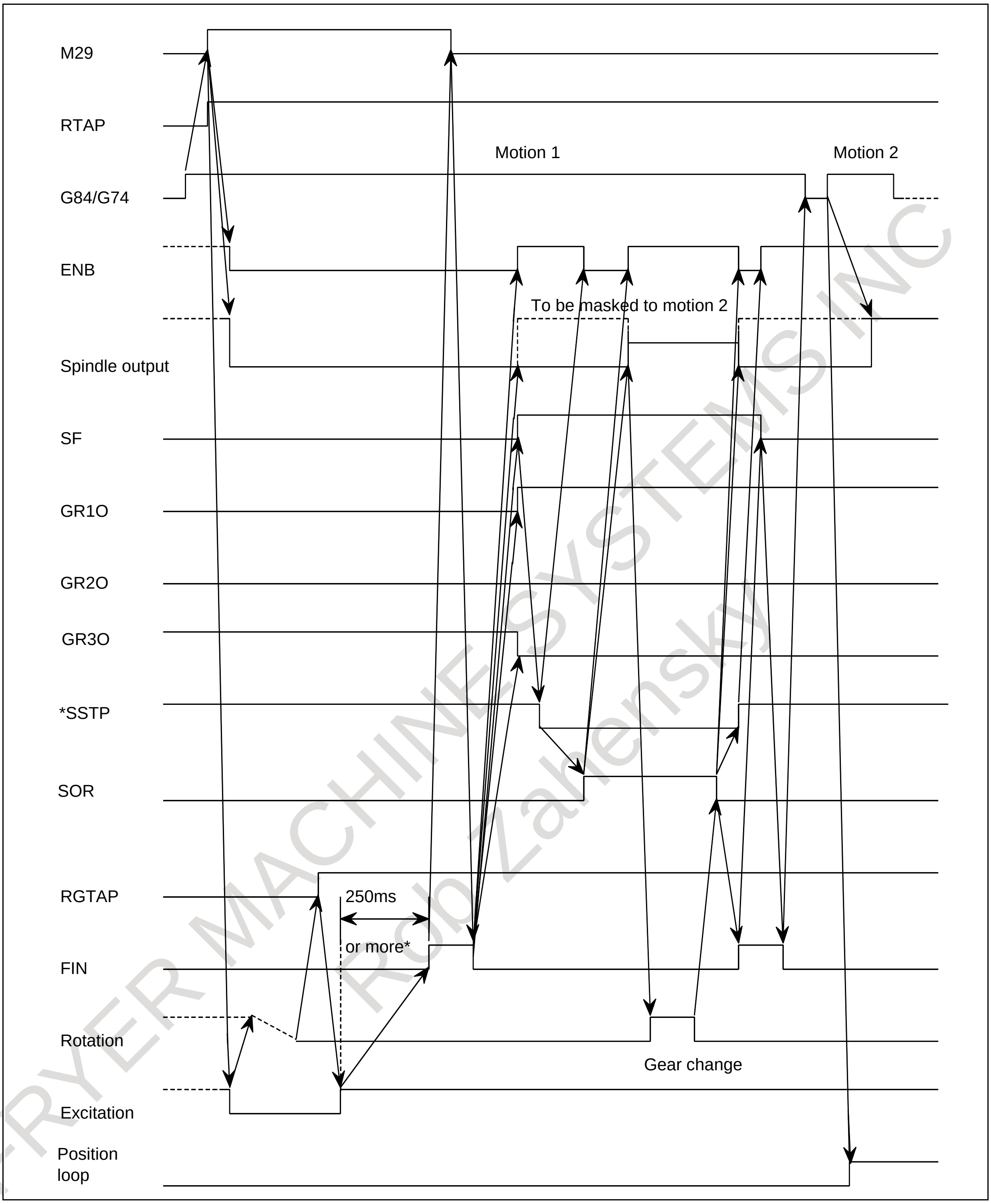


Fig. 10.13.15.3 (b) When gear change is performed (high to low gear)

NOTE

1 This time chart shows an example where the gear has shifted from high to low gear. One of the gear select signals (GR1O, GR2O, GR3O) has turned from "1" to "0", and one of the two remaining signals has turned from "0" to "1". This changes the gear.

2 The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

- T type gear selection method

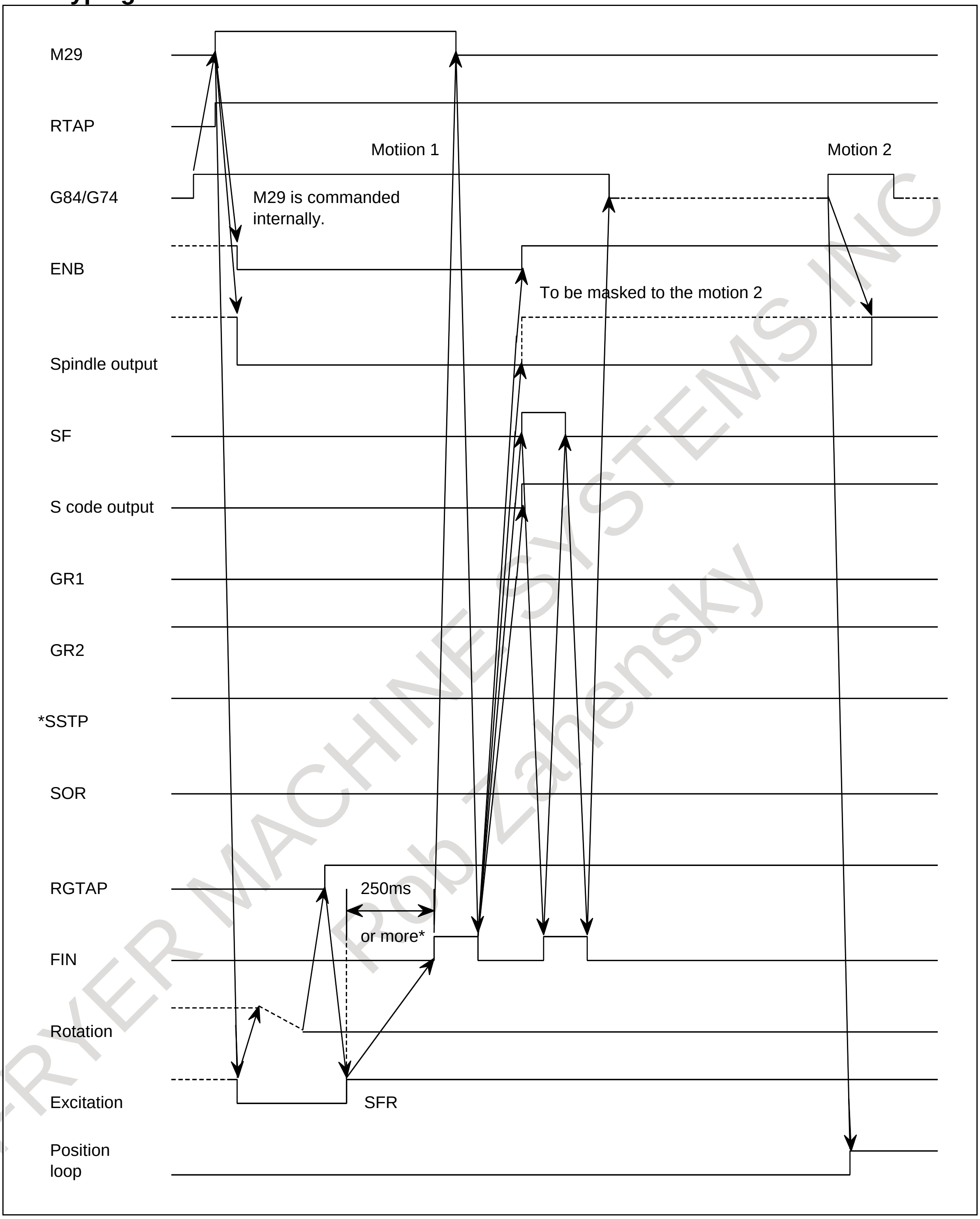


Fig. 10.13.15.3 (c) When gear change is not performed

> **NOTE**
> The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

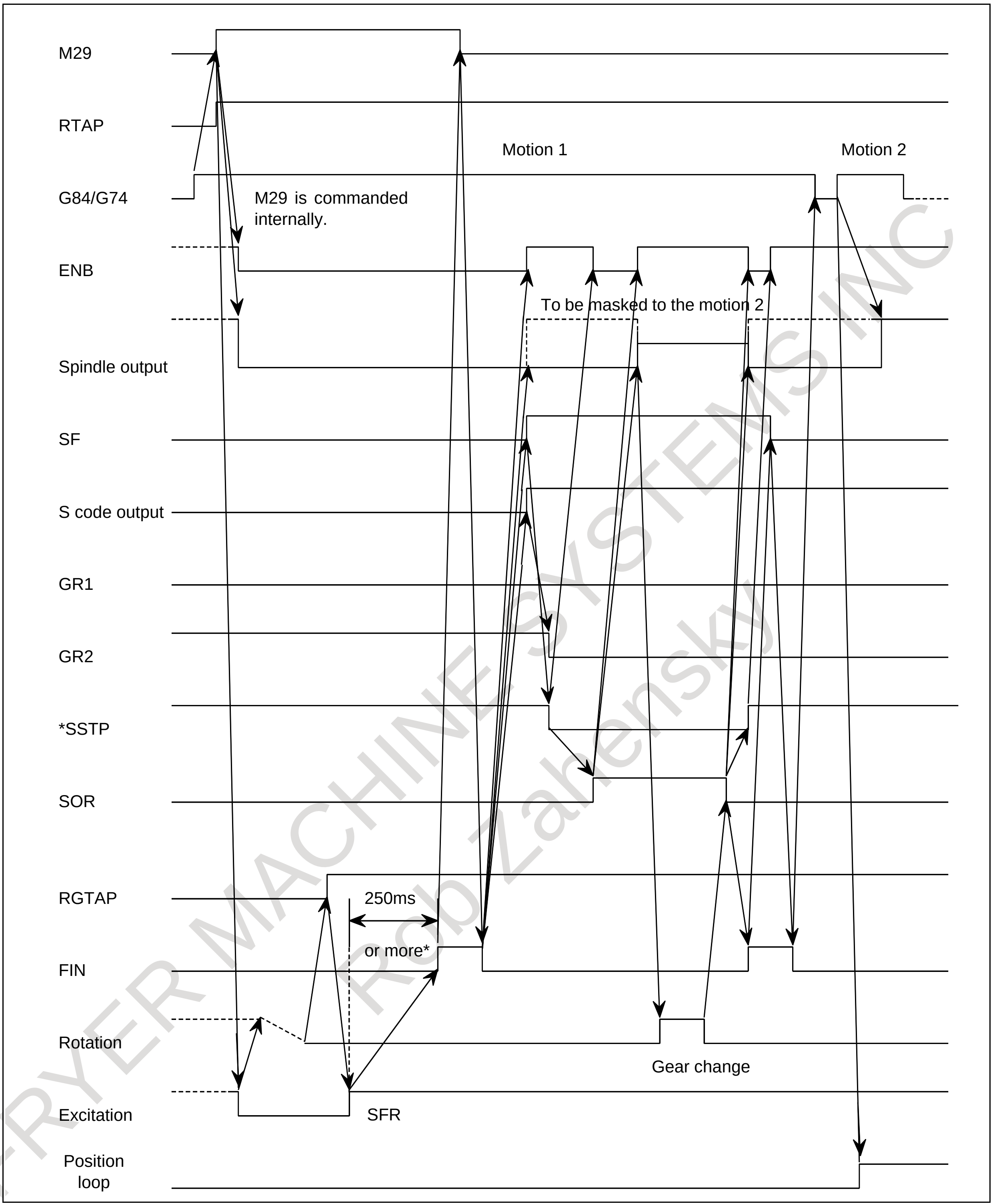


Fig. 10.13.15.3 (d) When gear-change is performed (high to low gear)

> **NOTE**
> 1 This time chart shows an example where the gear has changed from high to low gear. The PMC selects a required gear through an S code output and inputs the selected gear using the GR1 and GR2 signals to inform CNC of the selected gear.
> 2 The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

10.13.15.4 When M29 is specified before G84/G88

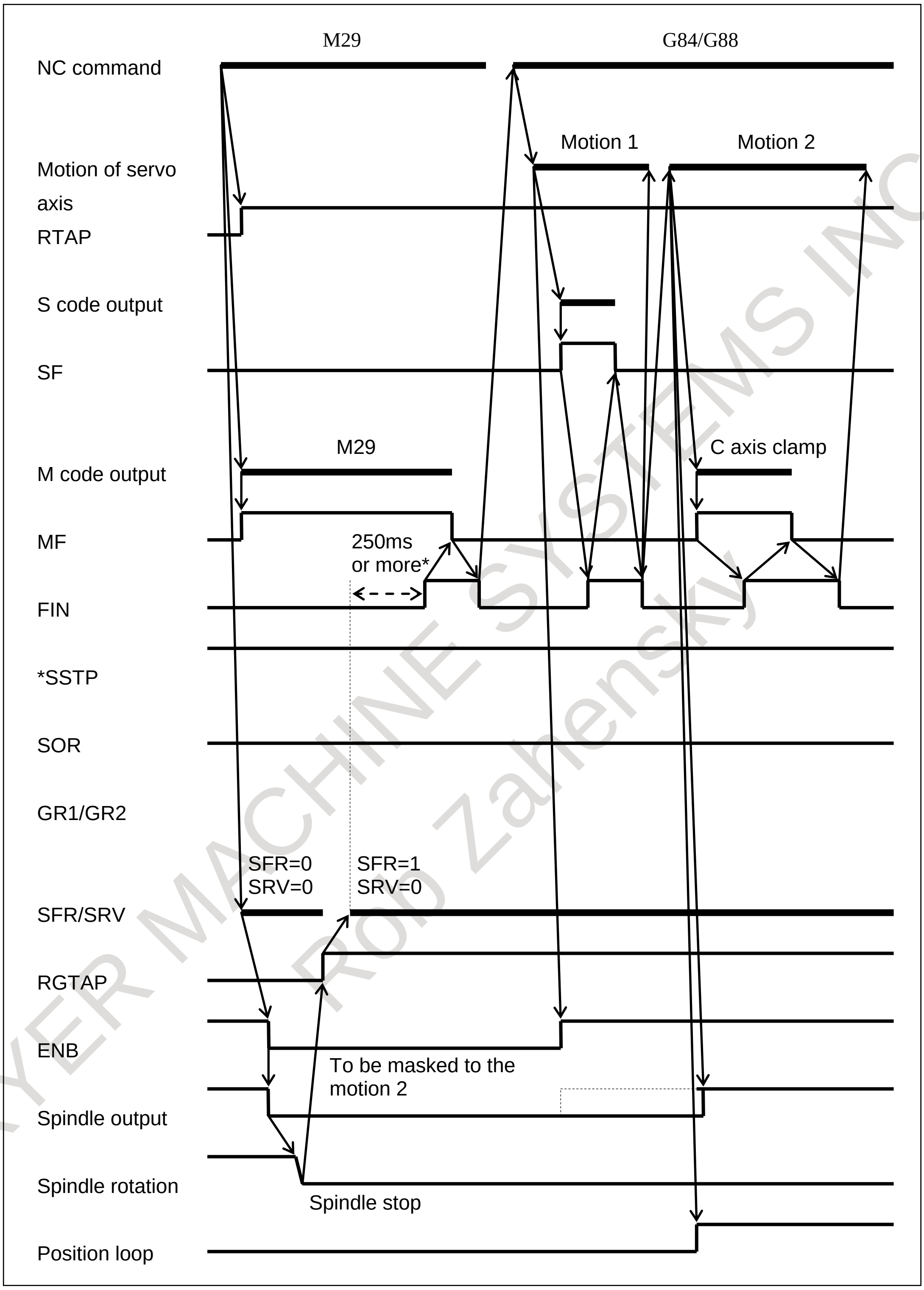


Fig. 10.13.15.4 (a) When gear change is not performed

> **NOTE**
> The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

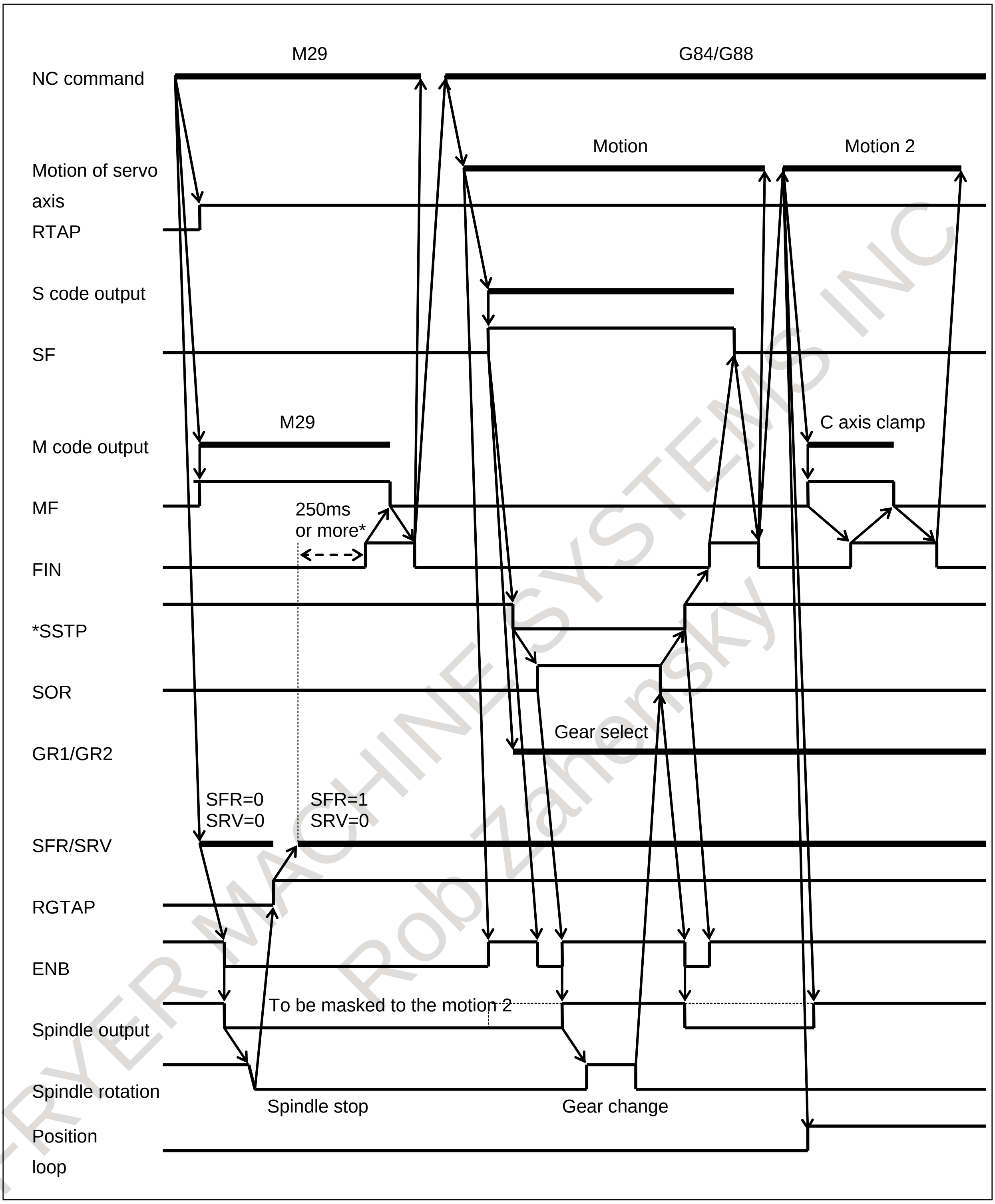


Fig. 10.13.15.4 (b) When gear change is performed

> **NOTE**
> The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

10.13.15.5 M29 and G84/G88 are specified in the same block

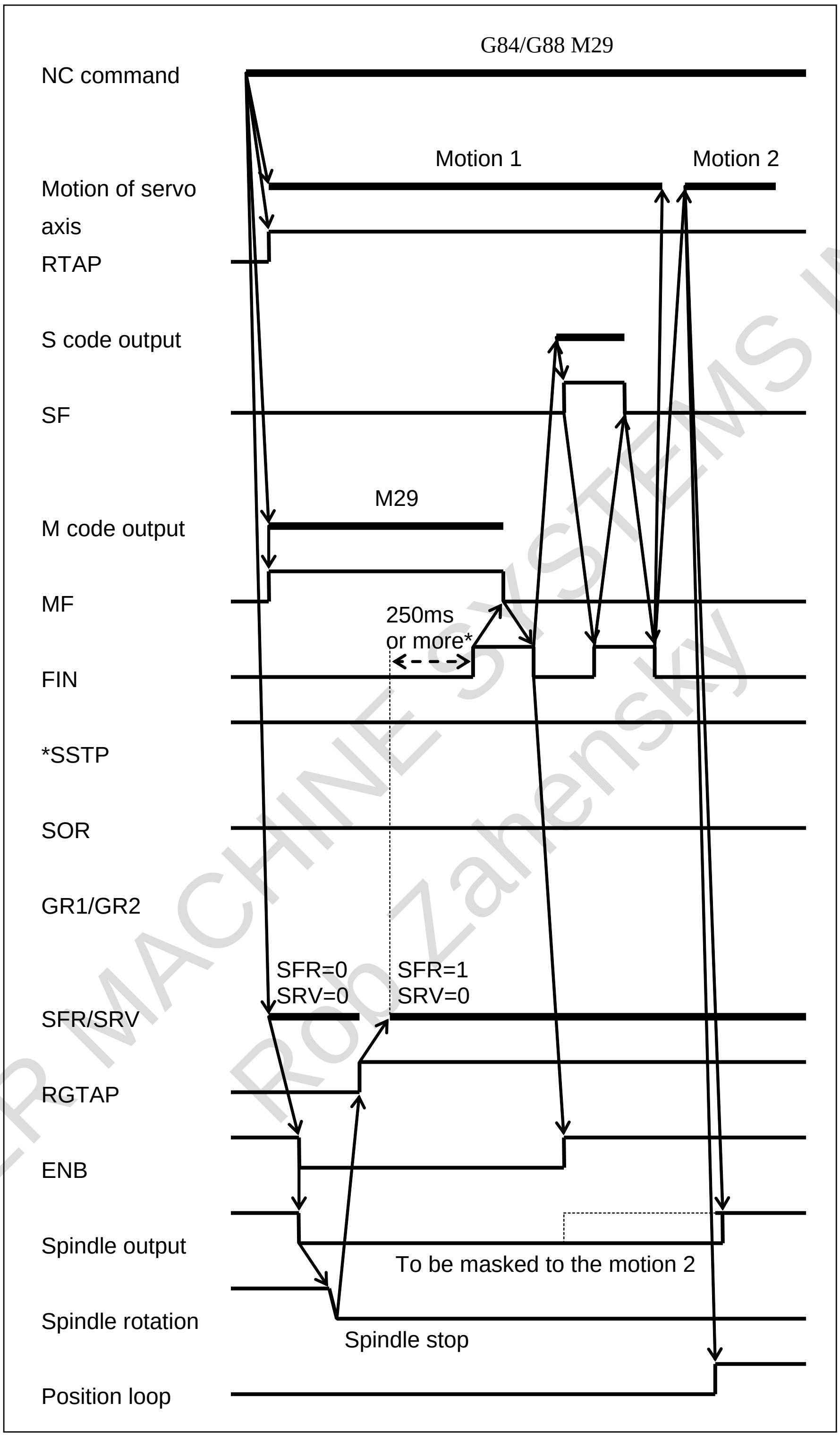


Fig. 10.13.15.5 (a) When gear-change is not performed

> **NOTE**
> The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

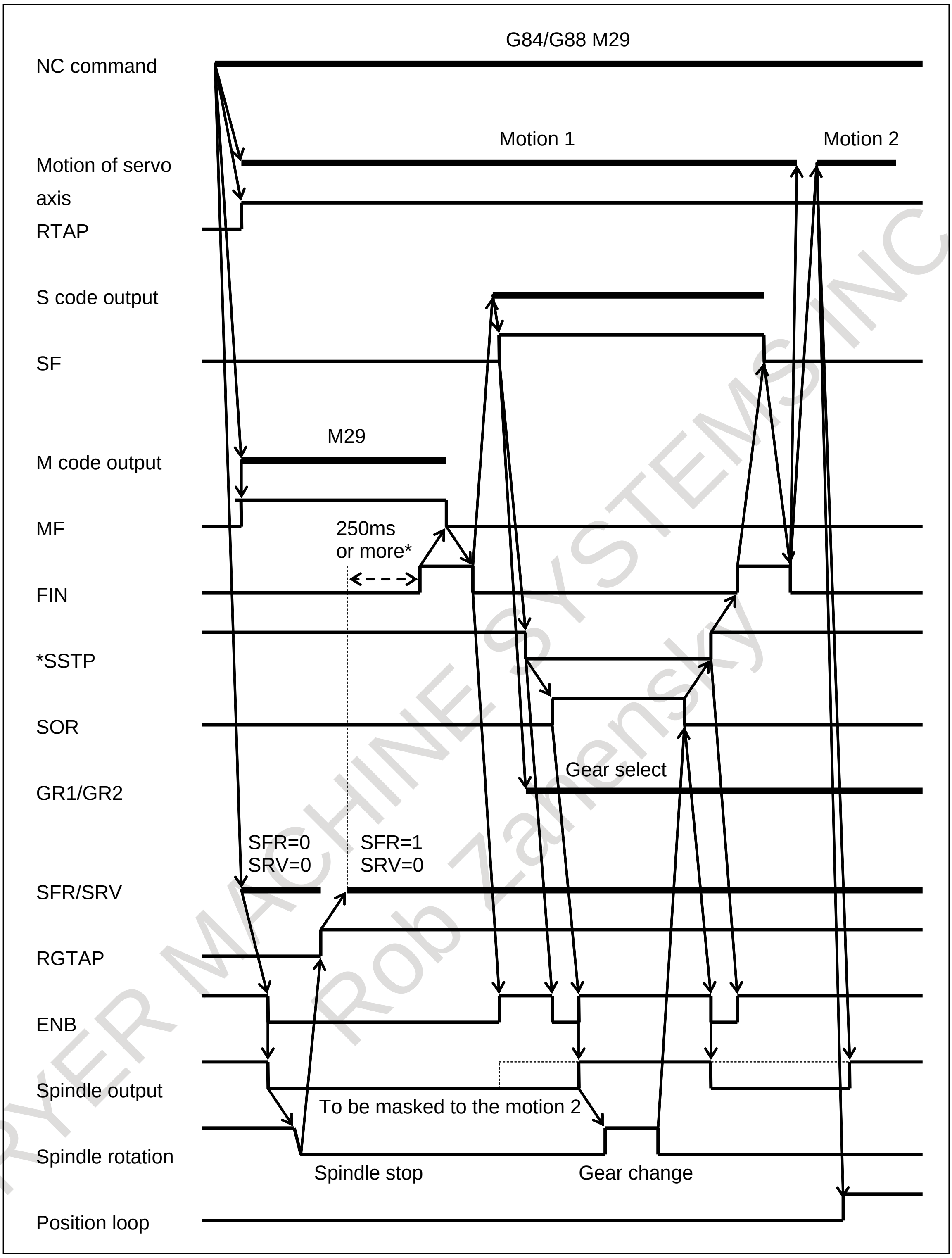


Fig. 10.13.15.5 (b) When gear-change is performed

> **NOTE**
> The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

10.13.15.6 Specifying G84/G88 for rigid tapping by parameters

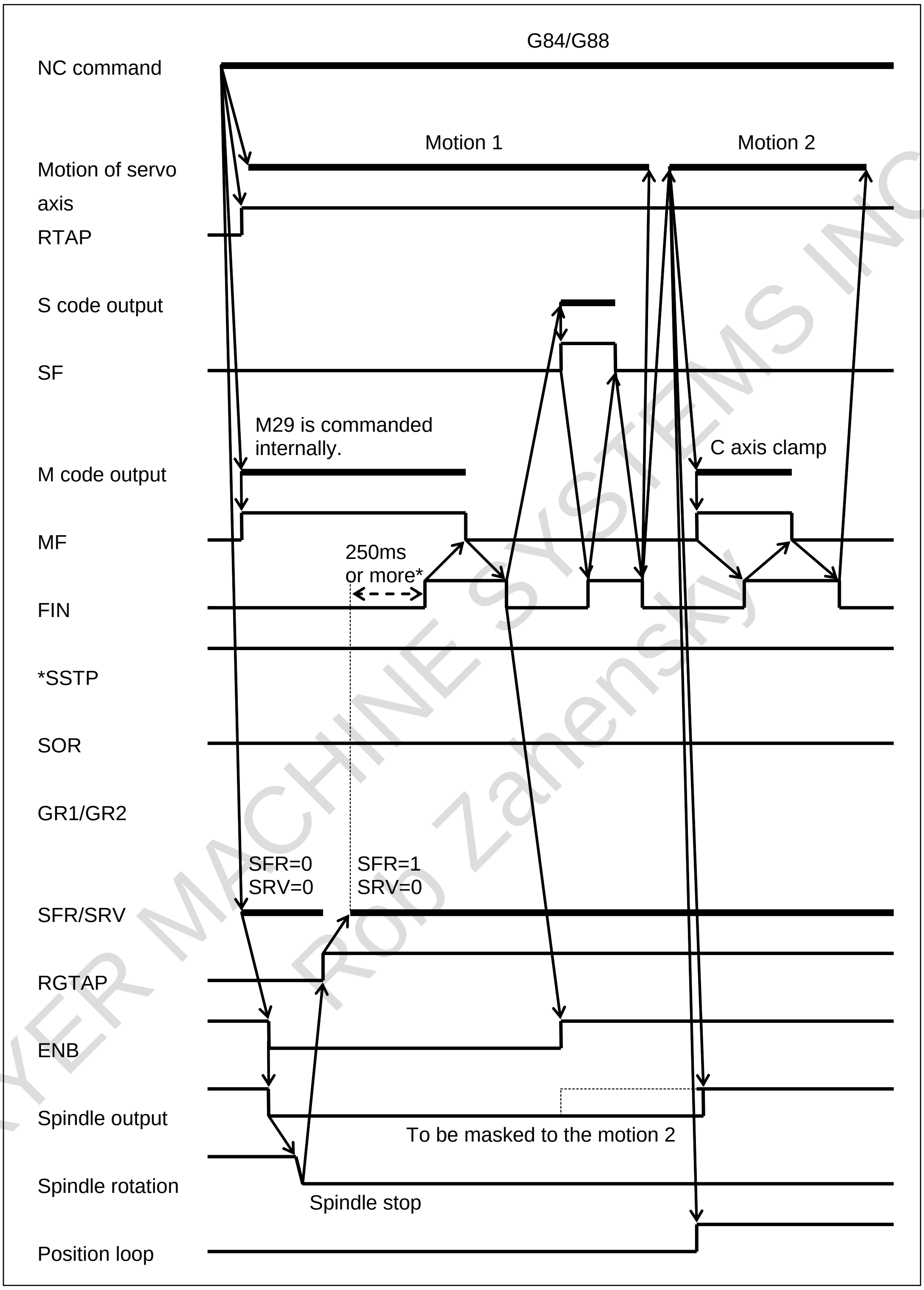


Fig. 10.13.15.6 (a) When gear-change is not performed

> **NOTE**
> The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

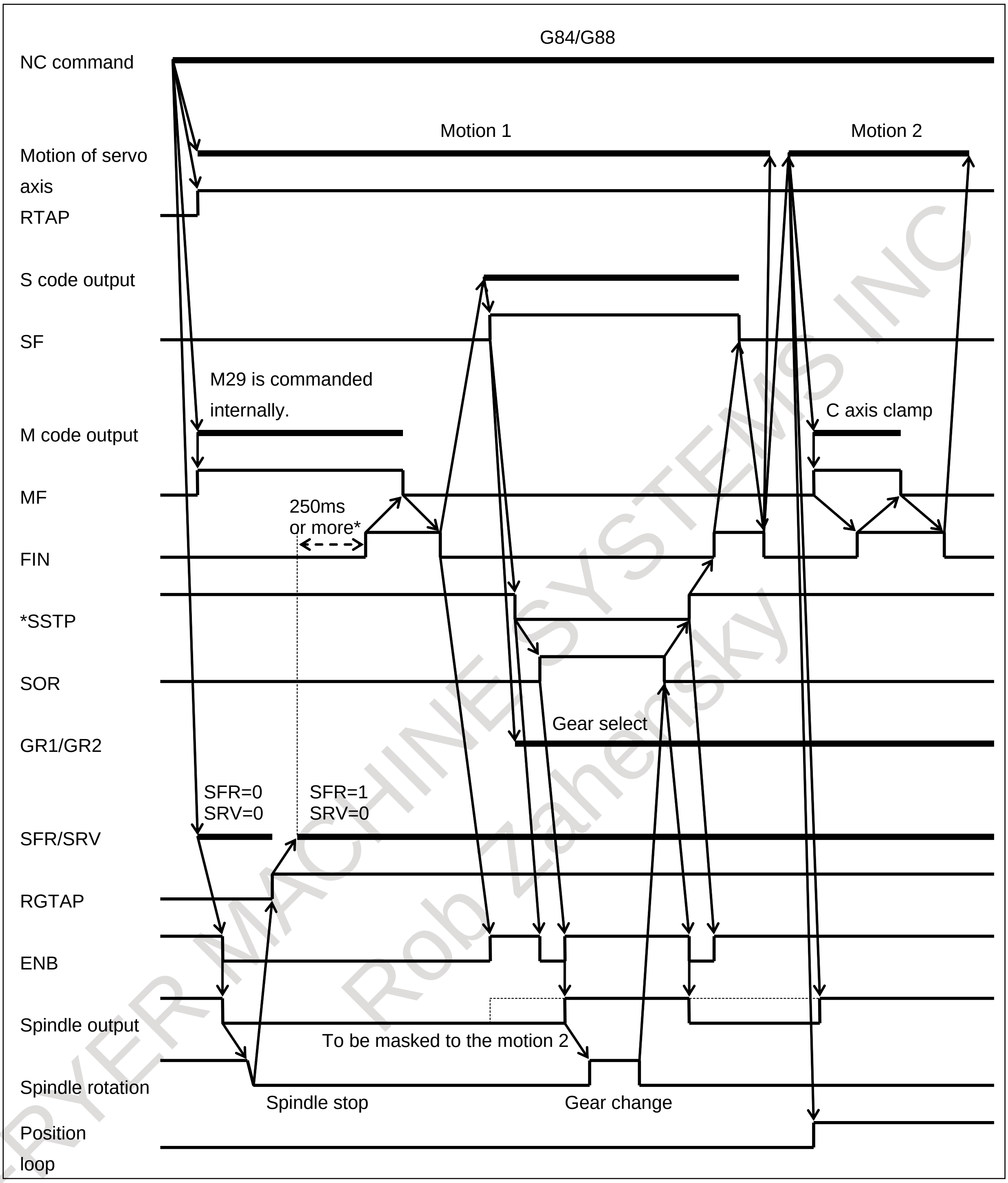


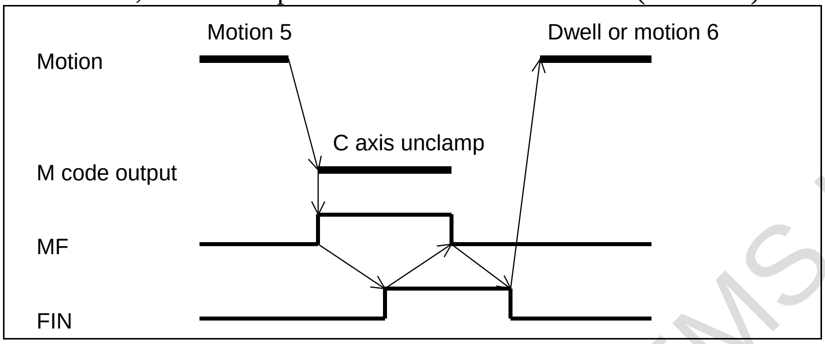
Fig. 10.13.15.6 (b) When gear-change is performed

NOTE
The delay(*) is not necessary, when an appropriate value is set to the parameter No.4099 for the delay time for stable motor excitation.

10.13.15.7 Timing of the M code for unclamping

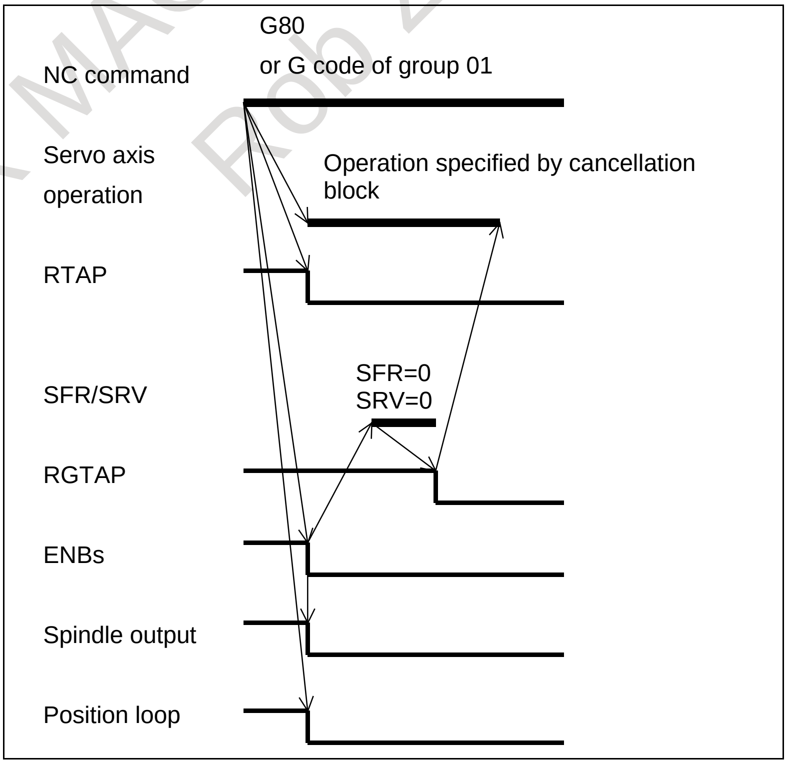
T

After extraction from the hole bottom to the R point level (motion 5), the M code for unclamping is output. When FIN is returned, dwell or rapid traverse to the initial level (motion 6) starts.



10.13.15.8 Timing to cancel rigid tapping mode

When rigid tapping is completed, the mode is canceled if a G code (such as G80, canned cycle G code, or Group 01 G code) is issued.
Spindle output has the effect of execution of S0. On the falling edge of the ENB signal (corresponding ENBs signal in multi-spindle control), cancel the rigid tapping mode of the PMC (for example, by turning off spindle activation) and set the rigid tapping signal to "0". After confirming that the rigid tapping signal is set to "0", proceed to the next block.
When gear change is performed using *SSTP and SOR, the ENB signal can be either "1" or "0". Do not cancel the PMC's rigid tapping mode at the falling edge of the ENB signal under these circumstances. The position loop is also canceled.
When the CNC is reset, the PMC's rigid tapping mode must be canceled.
When bit 2 (CRG) of parameter No. 5200 is 1, the system goes directly to the next block without checking that the rigid tapping signal is "0". Set CRG to 1 for systems in which the rigid tapping signal is always "1".



⚠ **CAUTION**

1 If rigid tapping mode is canceled by a Group 01 G code, such as G00 or G01, the block containing the G code is executed at the same time the ENB signal is turned to "0". Therefore, if the block contains an M code for controlling the spindle, an error may occur during processing in the PMC.

2 When bit 2 (CRG) of parameter No. 5200 is 1, if the next block contains an M code for controlling the spindle, an error may occur during processing in the PMC, when:
  - Rigid tapping mode is canceled by issuing G80
  - Rigid tapping mode is canceled by issuing a Group 01 G code , such as G00 or G01

NOTE

Rigid tapping mode is canceled as described above regardless of the gear selection method of M-type or T-type.

10.13.16 FSSB High-speed Rigid Tapping

Outline

By the FSSB communication between a CNC and a spindle amplifier, rotational position information of a spindle can be transmitted from a spindle amplifier to a servo control on CNC. The FSSB high-speed rigid tapping is a new function that a servo axis follows a spindle axis by utilizing this FSSB communication. And it is a characteristic of this function that the synchronization error becomes better than the ordinary function. And a synchronous error does not become worse even if a spindle error becomes worse by setting a shorter time constant for acceleration/deceleration than that of the former rigid tapping function. Therefore the machining cycle time can be shortened with keeping the synchronous accuracy.

In the following rigid tapping, this function cannot be used.

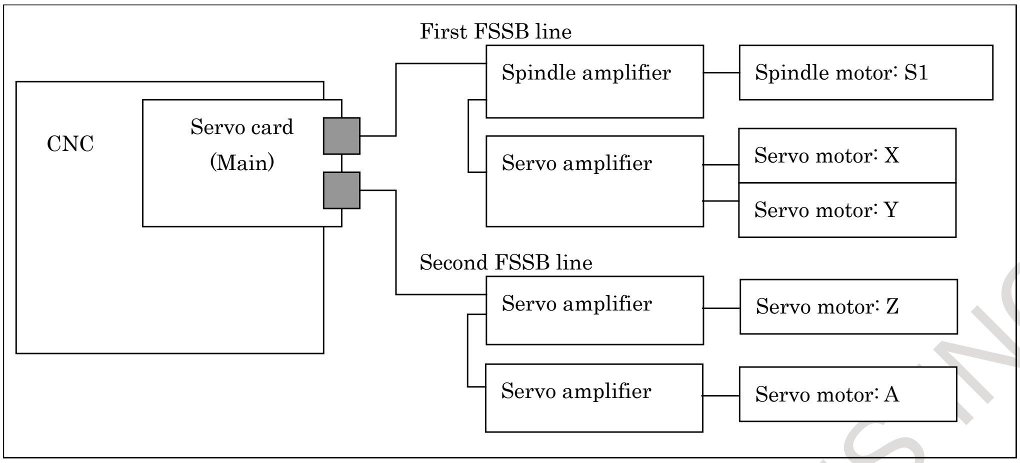
- The rigid tapping by the analog spindle or the spindle control with servo motor.
- The rigid tapping with one servo motor and multiple spindle motors by spindle command synchronous control.

Explanation

The spindle amplifier and the servo amplifier used for FSSB high-speed rigid tapping need to be connected to the main servo card. The spindle amplifier and the servo amplifier used for this function can be connected to both the first FSSB line and the second FSSB line.

Ex.1) FSSB high-speed rigid tapping with Z axis and S1 spindle axis in the one path system

[Example of connection of FSSB]



[Example of parameter setting]

When the bit 0 (FHR) of parameter No.24203 is set to 1, FSSB high-speed rigid tapping is enabled.

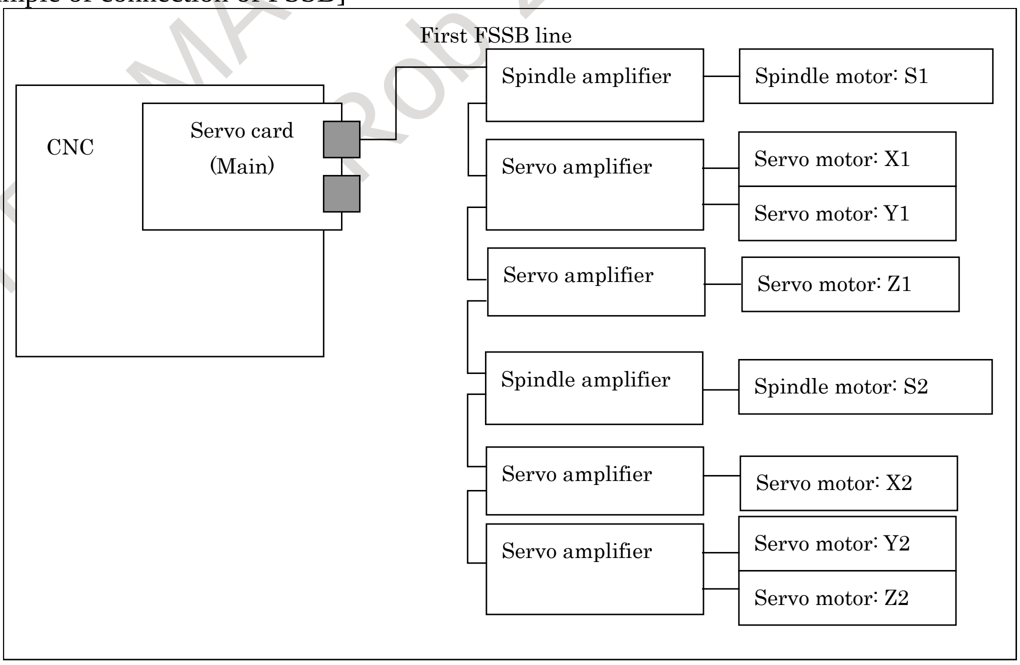
And, the index number of the spindle axis that synchronizes with the servo axis used by FSSB high-speed rigid tapping is set to the parameter (No.24204) as follows.

24204	SP INDEX OF SV FSSB SYNC
X	0
Y	0
Z	1
A	0

In addition, set 1 to the bit 1 (FHRSV) of parameter No.2429 and bit 1 (FEED) of parameter No.2005 of Z axis and the bit 1 (FHRSP) of parameter No.4549 of S1 spindle axis.

Ex.2) Two groups of FSSB high-speed rigid tapping with Z1 - S1 and Z2 - S2 in two paths system

[Example of connection of FSSB]



[Example of parameter setting]

When the bit 0 (FHR) of parameter No.24203 is set to 1, FSSB high-speed rigid tapping is enabled.

And, the index number of the spindle axis that synchronizes with the servo axis used by FSSB high-speed rigid tapping is set to the parameter (No.24204) as follows.

Path 1

24204	SP INDEX OF SV FSSB SYNC
X1	0
Y1	0
Z1	1

Path 2

24204	SP INDEX OF SV FSSB SYNC
X2	0
Y2	0
Z2	2

In addition, set 1 to the bit 1 (FHRSV) of parameter No.2429 and bit 1 (FEED) of parameter No.2005) of Z1 and Z2 axes and the bit 1(FHRSP) of parameter No.4549 of S1 and S2 spindle axes.

Eight or less index numbers of the spindle axis can be set in all system. If nine or more index numbers of the spindle axis or illegal index number of the spindle axis is set , the alarm PW0037, "SV/SP COMBINATION ERROR" is issued.

NOTE

1 Adjust precision and machining time by setting of the time constant for acceleration/deceleration in rigid tapping. In the ordinal rigid tapping, if the time constant for acceleration/deceleration is extremely short, the spindle motor cannot follow to the move command and the synchronization error of rigid tapping increases because the torque command of the spindle motor is saturated in the high speed rotation area. In the FSSB high-speed rigid tapping, even if the setting of the time constant for acceleration/deceleration is short up to the extent with which the torque command of the spindle motor is somewhat saturated, the synchronization precision can be maintained because the servo motor can follow to the rotation of the spindle motor. However, please set within the range where the overshoot is not occurs in the hole bottom.
Refer to the following manual for the details of the tuning way.
FANUC AC SPINDLE PARAMETER MANUAL (B-65280EN)
"Tuning way for FSSB High-speed Rigid Tapping"

2 When the parameter is changed, the power must be turned off before an operation is continued.

3 FSSB high-speed rigid tapping cannot be used with the rigid tapping by the analog spindle or the spindle control with servo motor. If the above rigid tapping is set, the alarm PW0037, "SV/SP COMBINATION ERROR" is issued.

4 FSSB high-speed rigid tapping cannot be used with the rigid tapping with a servo motor and multiple spindle motors by spindle command synchronous control. If the rigid tapping is commanded on the above condition, the alarm PS0203, "PROGRAM MISS AT RIGID TAPPING" is issued.

> **NOTE**
> 5 When FSSB high-speed rigid tapping is used with 3-dimensional rigid tapping, three axes of 3-dimensional rigid tapping need to set the same motor number of the spindle to the parameter (No.24204). If there is an axis whose index number of the spindle axis is different, the alarm PS0203, "PROGRAM MISS AT RIGID TAPPING" is issued on the command of rigid tapping. In addition, three axes of 3-dimensional rigid tapping need to set the bit 1 (FHRSV) of parameter No.2429 and bit 1 (FEED) of parameter No.2005.
> 6 When Peck rigid tapping cycle is used (bit 5 (PCP) of parameter No.5200 is 1), set 1 to the bit 7 (PRA) of parameter No.5209. If the illegal program that the depth of cut is smaller than the cutting start distance is commanded, the alarm PS5560, "ILLEGAL DEPTH OF CUT" can be issued.
> 7 The setting of the optional gear ratio cannot be used.
> 8 If the exchanged axes by flexible path axis assignment or composite control are used by FSSB high-speed rigid tapping, the motor number of the spindle axis that synchronizes with each servo axis (the parameter (No.24204)) need to set. In addition, set 1 to bit1 (FHRSV) of parameter No.2429 and Bit 1 (FEED) of parameter No.2005.
> 9 If the tapping axis of FSSB high-speed rigid tapping is the master axis of axis synchronous control, the same motor number of the spindle (the parameter (No.24204)) need to be set to the slave axes. In FSSB high-speed rigid tapping, synchronization establishment and synchronization error compensation cannot be used.
> 10 If the tapping axis of FSSB high-speed rigid tapping is the master axis of tandem control, it is not necessary to set the same motor number of the spindle (the parameter (No.24204)) to the slave axes.
> 11 With FSSB high-speed rigid tapping, the following servo software functions cannot be used:
> - Dual position feedback function
> If FSSB high-speed rigid tapping is used with either of the above functions, the alarm SV0417, "ILL DGTL SERVO PARAMETER" (Diagnosis data No.4291) is issued.
> 12 With this function, spindle differential speed control cannot be used.
> 13 All of CNC, servo, and spindle software need to support this function. If even one of the software doesn't support it, the ordinal rigid tapping is executed or the excess error alarm is issued.

Limitation

Synchronous control (Synchronous/Composite control)

An axis of synchronous control cannot be used as a tapping axis for FSSB high-speed rigid tapping.

Angular axis control

An angular axis cannot be used as a tapping axis for rigid tapping.

Diagnosis Data

1612	The index number of the spindle axis that synchronizes with each servo axis

[Data Type] Byte axis

The index number of the spindle axis that can use direct communication between a spindle amplifier and a servo amplifier on FSSB connection is displayed.

10.13.16.1 Change the combination for FSSB high-speed rigid tapping

Outline

In FSSB high-speed rigid tapping, the combination of spindle axis and servo axis to synchronize can be changed after power-on.

Explanation

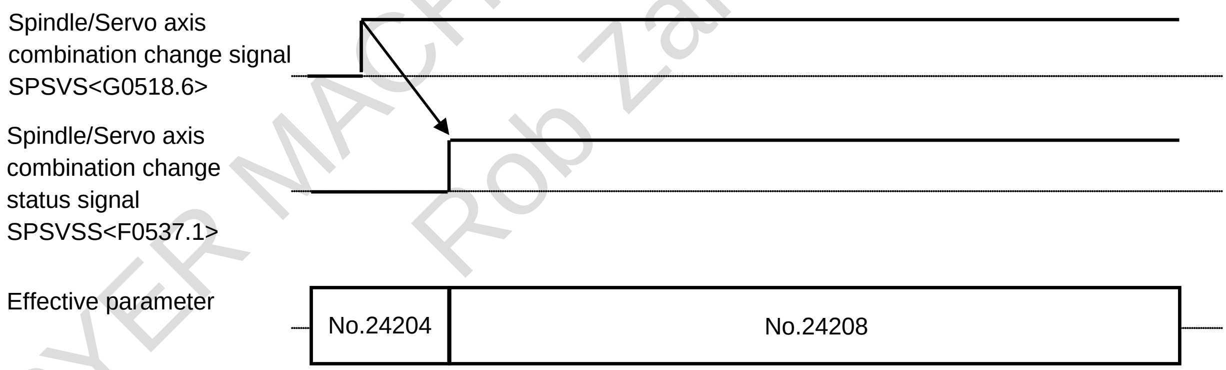
This function makes it possible to choose whether to use parameter No.24204 or No.24208 by PMC input signals to change the combination of spindle axis and servo axis to synchronize.

Spindle/Servo axes combination change (Parameter change from No.24204 to No.24208)

Set Spindle/Servo axis combination change signal SPSVS<G0518.6> from “0” to “1”. After the change of spindle/servo axes combination completes, Spindle/Servo axis combination change status signal SPSVSS<F0537.1> is set from “0” to “1”.

> **NOTE**
> The alarm (DS0092) “SV/SP COMBINATION ERROR” issues in the following cases.
> - Parameter No.24208 is an illegal setting.
> - The function using parameter No.24204 (FSSB high-speed rigid tapping, Servo/Spindle synchronous control (FSSB type), Electronic gear box (FSSB type)) is being applied.
> - Rigid tapping, Servo/Spindle synchronous control or Electronic gear box is being applied in the path that uses parameter No.24208.

Time chart

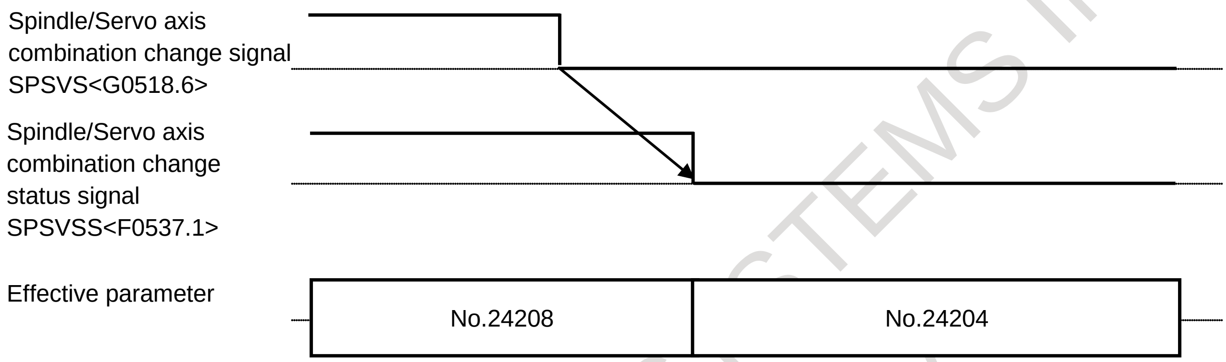


Spindle/Servo axes combination change (Parameter change from No.24208 to No.24204)

Set Spindle/Servo axis combination change signal SPSVS<G0518.6> from “1” to “0”. After the change of spindle/servo axes combination completes, Spindle/Servo axis combination change status signal SPSVSS<F0537.1> is set from “1” to “0”.

> **NOTE**
> The alarm (DS0092) “SV/SP COMBINATION ERROR” issues in the following cases.
> - The function using parameter No.24208 (FSSB high-speed rigid tapping, Servo/Spindle synchronous control (FSSB type), Electronic gear box (FSSB type)) is being applied.
> - Rigid tapping, Servo/Spindle synchronous control or Electronic gear box is being applied in the path that uses parameter No.24204.

Time chart

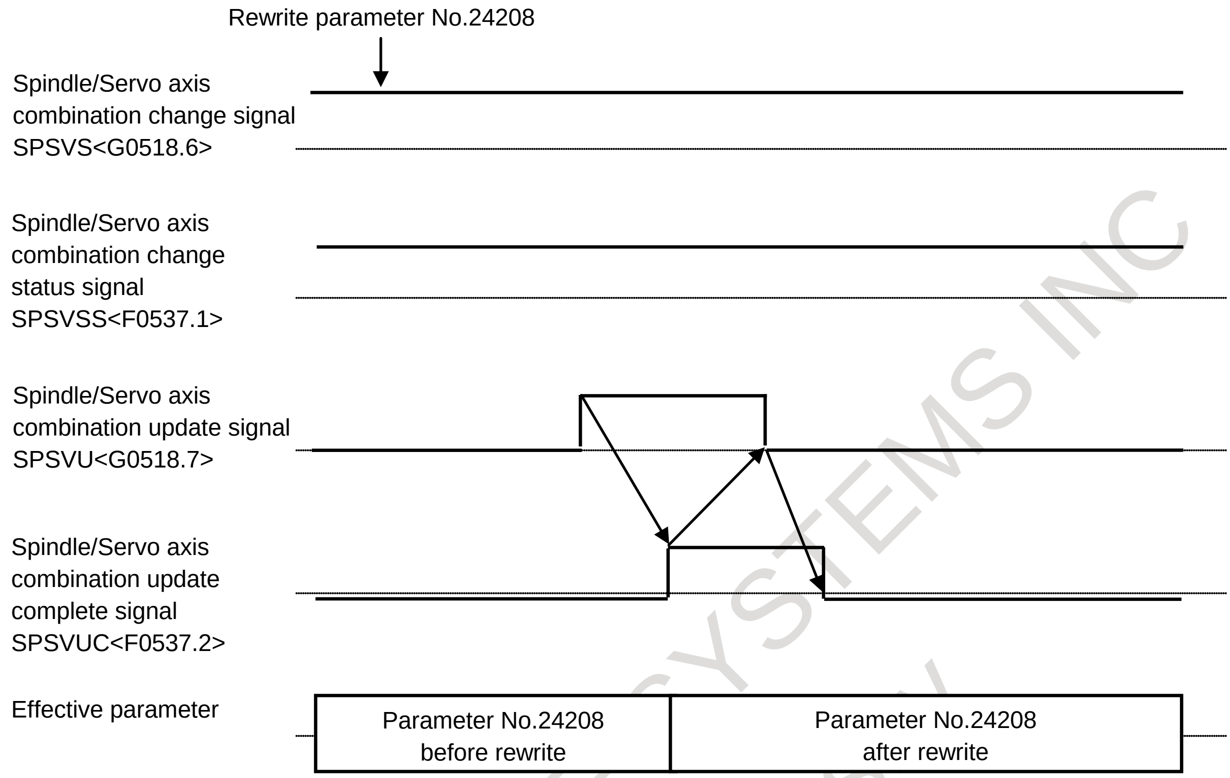


Spindle/Servo axes combination update (Update of parameter No.24208)

Set Spindle/Servo axis combination update signal SPSVU<G0518.7> from “0” to “1” with Spindle/Servo axis combination change status signal SPSVSS<F0537.1> equal to “1”. After the update of spindle/servo axes combination completes, Spindle/Servo axis combination update complete signal SPSVUC<F0537.2> is set to “1”.

> **NOTE**
> The alarm (DS0092) “SV/SP COMBINATION ERROR” issues in the following cases.
> - Parameter No.24208 is an illegal setting.
> - The function using parameter No.24208 (FSSB high-speed rigid tapping, Servo/Spindle synchronous control (FSSB type), Electronic gear box (FSSB type)) is being applied.
> - Rigid tapping, Servo/Spindle synchronous control or Electronic gear box is being applied in the path that uses parameter No.24208.

Time chart



Signal

Spindle/Servo axis combination change signal SPSVS<G0518.6>

[Classification] Input signal

[Function] This signal changes the combination of spindle axis and servo axis to synchronize.

[Operation] When setting this signal from "0" to "1", the parameter to set the combination of spindle axis and servo axis to synchronize is changed from No.24204 to No.24208.
When setting this signal from "1" to "0", the parameter to set the combination of spindle axis and servo axis to synchronize is changed from No.24208 to No.24204.

Spindle/Servo axis combination update signal SPSVU<G0518.7>

[Classification] Input signal

[Function] This signal updates the combination of spindle axis and servo axis to synchronize (Parameter No.24208).

[Operation] When setting this signal from "0" to "1", the combination of spindle axis and servo axis to synchronize (Parameter No.24208) is updated. This signal is valid when Spindle/Servo axis combination change status signal SPSVSS<F0537.1> is equal to "1".

Spindle/Servo axis combination change status signal SPSVSS<F0537.1>

[Classification] Output signal

[Function] This signal indicates the status of the change in the combination of spindle axis and servo axis to synchronize.

[Output condition] This signal is set to "1" in the following case:
- When the parameter to set the combination of spindle axis and servo axis to synchronize is changed to No.24208 by Spindle/Servo axis combination change signal SPSVS<G0518.6>.
This signal is set to "0" in the following case:

- When the parameter to set the combination of spindle axis and servo axis to synchronize is changed to No.24204 by Spindle/Servo axis combination change signal SPSVS<G0518.6>.
- When the combination of spindle axis and servo axis to synchronize is not changed by Spindle/Servo axis combination change signal.

Spindle/Servo axis combination update complete signal SPSVUC<F0537.2>

[Classification] Output signal

[Function] This signal indicates that the combination of spindle axis and servo axis to synchronize (Parameter No.24208) has completed being updated.

[Output condition] This signal is set to “1” in the following case:

- When the combination of spindle axis and servo axis to synchronize (Parameter No.24208) is updated by Spindle/Servo axis combination update signal SPSVU<G0518.7>.

This signal is set to “0” in the following case:

- When Spindle/Servo axis combination update signal SPSVU<G0518.7> is set to “0”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0518	SPSVU	SPSVS						

	#7	#6	#5	#4	#3	#2	#1	#0
F0537						SPSVUC	SPSVSS	

Notes

- When the alarm (DS0092) "SV/SP COMBINATION ERROR" is issued, the combination of spindle axis and servo axis is not updated and remains in the last status.
- When controlling Spindle/Servo axis combination change signal SPSVS<G0518.6> or Spindle/Servo axis combination update signal SPSVU<G0518.7> during operation, use M codes that do not perform Buffering.
- Parameter No.24204 can be replaced with parameter No.24208 by this function when applying the functions that use parameter No.24204, other than FSSB high-speed rigid tapping.
- When applying this function, check the ready status with All-spindle operation ready signal SRSRDY<F0034.7> after power-on.

10.13.17 Smart Rigid Tapping

Outline

Smart rigid tapping is the tapping function that uses maximum spindle power in all speed range. As the servo axis follows spindle axis with using FSSB communication, the synchronous error is small.
The fastest rigid tapping can be achieved with no parameter tuning basically.

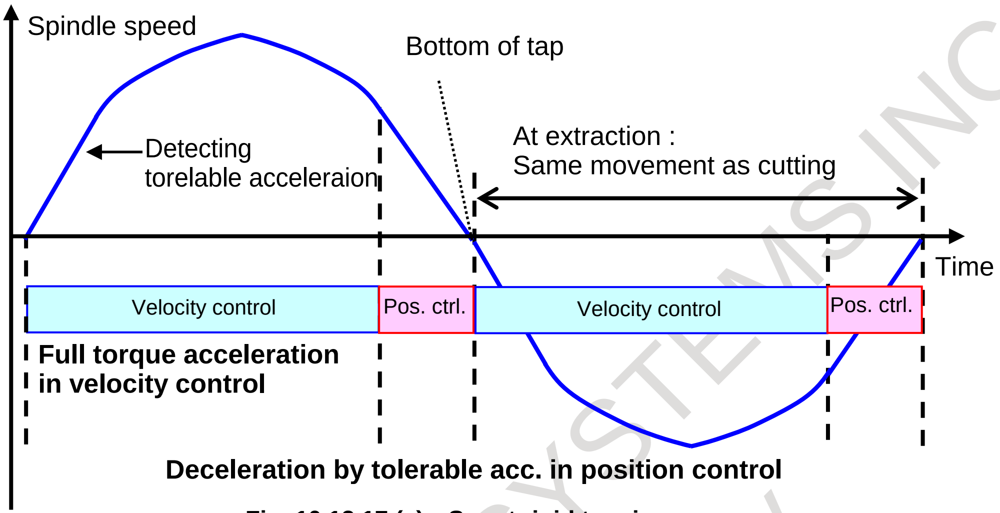


Fig. 10.13.17 (a) Smart rigid tapping

Application example of smart rigid tapping (Pitch 1mm, Length25mm)

Before applying Smart rigid tapping

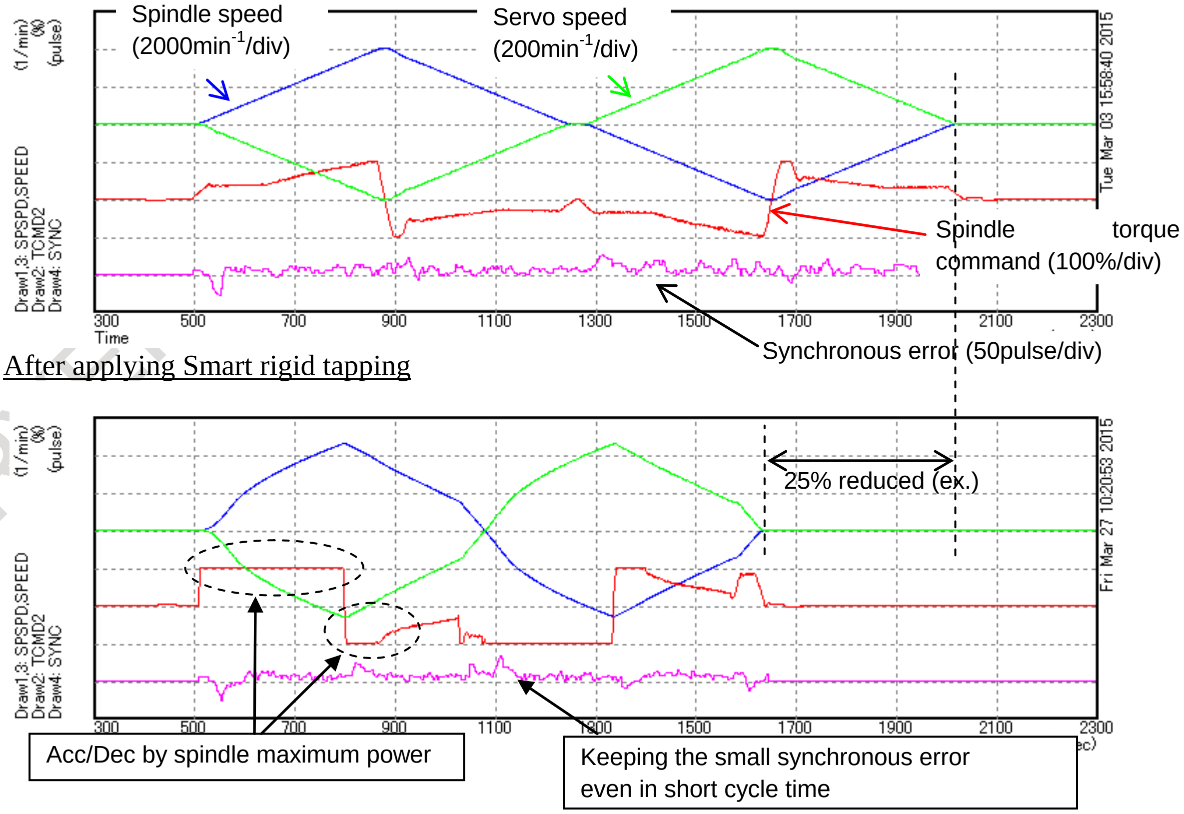


Fig. 10.13.17 (b) Application example of smart rigid tapping

Program Format

The program format for Smart rigid tapping is same as ordinal rigid tapping. But if the moving distance of tapping is larger than over 4096 times of screw pitch, the alarm PS0564, "ILLEGAL COMMAND OF SMART RTAP" will occur.

DI/DO Signals and Sequence

The DI/DO signals and PMC sequence for ordinal rigid tapping can be used without any change. There is no special DI/DO signals and PMC sequence for Smart rigid tapping.
Regarding DI/DO signals and PMC sequence, refer to the chapter of "Signal" and "Timing Charts for Rigid Tapping Specification" in this manual.

Parameters

Settings for FSSB high-speed rigid tapping

Set all parameters related to FSSB high-speed rigid tapping so that the servo axis follows spindle axis. The detail of setting parameters are shown in the chapter of "FSSB high-speed rigid tapping" and "Parameter" in this manual.

Table 10.13.17 (a) Settings for FSSB high-speed rigid tapping

Parameter No.	Contents	Setting value
2005#1	Feed-forward function bit (Servo)	1
2429#1	FSSB high-speed rigid tapping function bit (Servo)	1
2610	Position gain of servo axis during FSSB high-speed rigid tapping	5000
4549#1	FSSB high-speed rigid tapping function bit (Spindle)	1
24203#0	FSSB high-speed rigid tapping function bit (CNC)	1
24204	The index number of the spindle axis that synchronizes to each servo axis	Depends on configuration

Settings for Smart rigid tapping

After setting of following parameters the settings for Smart rigid tapping will be finished in case that the settings for Normal rigid tapping has been already done. In case that the settings for Normal rigid tapping has not been done yet, do it with referring to chapter of "Parameter" in this manual, or the chapter of "Rigid tapping" in FANUC AC SPINDLE MOTOR α*i*/β*i* series BUILT-IN SPINDLE MOTOR B*i* series parameter manual (B-65280EN).

Table 10.13.17 (b) Settings for Smart rigid tapping

Parameter No.	Contents	Setting value
5206#1	Smart rigid tapping function bit	1
2423#2	Setting for the compensation of reverse operation in Smart rigid tapping	1
5214	Setting of an allowable rigid tapping synchronous error range	0
4105 4107	Time constant of magnetic flux for Magnetic flux control	Specific parameter for each motor
4661	Tolerable error value at spindle stop	0

Settings for Each Machine

As Smart rigid tapping accelerates by maximum spindle power, the tuning is not necessary basically. But there may be some cases that a shock occurs in servo axis which follows spindle acc./dec. by maximum power. In such case, use following parameters.

Parameters for tuning Smart rigid tapping

Table 10.13.17 (c) Parameters for tuning Smart rigid tapping

Parameter No.	Contents	Setting value
4666 4667	Bell shape acc./dec. time constant for Smart rigid tapping	0
4668 4669	Upper limit of torque command for Smart rigid tapping	0
4395#5	Improvement of deceleration for Smart rigid tapping	0
4790	Lower limit of torque command for Smart rigid tapping in position control	0
2611	Coefficient of velocity feed-forward for feed axis in Smart rigid tapping	0
2613	Compensation of reverse operation in Smart rigid tapping	0

Settings for Smart thermal control

As Smart rigid tapping uses maximum spindle power in tapping, there is a possibility that the motor overheat alarm (SP9001) might occur in severe duty tapping.
Smart thermal control uses the spindle motor power at the best without motor overheat.
Regarding the detail of Smart thermal control, refer to FANUC AC SPINDLE MOTOR α*i*/β*i* series BUILT-IN SPINDLE MOTOR B*i* series parameter manual (B-65280EN).

Settings for rigid tapping during dry run

Rigid tapping during dry run can be selected by bit 2 (NSD) of parameter No.5206 from Smart rigid tapping or ordinal FSSB high-speed rigid tapping. Set 1 to bit 2 (NSD) of parameter No.5206 which is described below if you make override effective during dry run.

Bit 2 (NSD) of parameter No.5206 is 0

Rigid tapping during dry run is Smart rigid tapping.

- Feedrate cannot be changed by manual feedrate override signal *JV0 to *JV15<Gn010, Gn011> or manual rapid traverse selection signal RT<Gn019.7>.
- If dry run is canceled during Smart rigid tapping in dry run, feedrate is maintained until finishing tapping.
- If dry run is switched during Smart rigid tapping, feedrate is maintained until finishing tapping.

Bit 2 (NSD) of parameter No.5206 is 1

Rigid tapping during dry run is ordinal FSSB high-speed rigid tapping.

- Feedrate can be changed by manual feedrate override signal *JV0 to *JV15<Gn010, Gn011> or manual rapid traverse selection signal RT<Gn019.7>.
- If dry run is canceled during FSSB high-speed rigid tapping in dry run, feedrate is changed from dry run feedrate to feedrate commanded in machining program.
- If dry run is switched during Smart rigid tapping, Smart rigid tapping is continued and feedrate is maintained until finishing tapping.
- Acc./dec. time constant setting is necessary because ordinal FSSB high-speed rigid tapping is executed during dry run. Adjustment of optimal time constant is unnecessary. Set relative long time constant such as 500[msec] to parameter No.5261 to No.5264. Refer to "Parameter list related to Normal rigid tapping" for detail.

Tuning of Rigid Tapping

Common used parameter list

The following parameter list show the common used parameters both for Normal rigid tapping and Smart rigid tapping. In case that the tuning for Normal rigid tapping has finished, it's not necessary to change these parameters. Regarding the detail of each parameter, refer to the chapter of "Parameter" in this manual or the chapter of "Rigid tapping" in FANUC AC SPINDLE MOTOR α*i*/β*i* series BUILT-IN SPINDLE MOTOR B*i* series parameter manual (B-65280EN).

Table 10.13.17 (d) Common used parameter list

Parameter No.	Contents
5210	Rigid tapping mode specification M code
5202#0	Whether to perform orientation (reference position return) when starting rigid tapping
5221 5222 5223 5224	Teeth number of spindle side at arbitrary gear ratio (command) setting (No. 5224 is used for the T series only.)
5231 5232 5233 5234	Teeth number of position coder side at arbitrary gear ratio (command) setting (No. 5234 is used for the T series only.)
5241 5242 5243 5244	Spindle maximum speed at rigid tapping (No.5244 is used for the T series only.)
5200#4	Override selection at extracting
5211	Override value at extracting
5300 5302 5304 5306	In-position width of tapping axis
5301	In-position width of spindle
4000#4	Reference position return direction on servo mode
4002#5	Whether to enable the rotation direction signal (SFR/SRV) on servo mode
4006#7	Setting of the command arbitrary gear ratio function (CMR) on rigid tapping
4017#7	Shortcut function when orientation is specified in stop state
4044 4045	Velocity loop proportional gain on servo mode (It is selected by input signal CTH1A/B.)
4052 4053	Velocity loop integral gain on servo mode (It is selected by input signal CTH1A/B.)
4056 to 4059	Gear ratio between spindle and motor (It is selected by input signal CTH1A or CTH2A)
4065 to 4068	Spindle position gain on servo mode/spindle synchronous control (It is selected by input signal CTH1A or CTH2A)
4073	Grid shift amount on servo mode
4074	Reference position return speed on servo mode
4091	Position gain change ratio at reference position return time on servo mode
4099	Delay time for stable motor excitation
4171 4173	Denominator of an arbitrary gear ratio between the motor sensor and spindle (It is selected by input signal CTH1A or CTH2A.)
4172 4174	Numerator of an arbitrary gear ratio between the motor sensor and spindle (It is selected by input signal CTH1A or CTH2A.)
4406	Acceleration/deceleration time constant for Cs contouring control/servo mode

Parameter list related to Normal rigid tapping

In case of the retraction of rigid tapping after the breakage of tool on cutting by Smart rigid tapping, Normal rigid tapping is necessary even if Smart rigid tapping. To use Normal rigid tapping, the setting of time constant parameter for acc./dec. is required. The following list shows the tuning parameter for Normal rigid tapping. In case that the tuning for Normal rigid tapping has finished, it's not necessary to change these parameters. Regarding the detail of each parameter, refer to the chapter of "Parameter" in this manual or the chapter of "Rigid tapping" in FANUC AC SPINDLE MOTOR α*i*/β*i* series BUILT-IN SPINDLE MOTOR B*i* series parameter manual (B-65280EN).

Table 10.13.17 (e) Parameter list related to Normal rigid tapping

Parameter No.	Contents
5280 5281 to 5284	Position gain of tapping axis at rigid tapping (No. 5284 is used for the T series only.)
5261 5262 5263 5264	Acc/Dec time constant (No. 5264 is used for the T series only.)
5201#2 5271 to 5274	Time constant for acceleration/deceleration in rigid tapping extraction (No. 5274 is used for the T series only.)
5203#2	Feed-forward function at rigid tapping
5310 5350 5354 5358	Allowable level of position error of tapping axis at moving
5311	Allowable level of position error of spindle at moving
5312 5352 5356 5360	Allowable level of position error of tapping axis at stop
5313	Allowable level of position error of spindle at stop
5203#5	Selection of bell shape acc./dec. for rigid tapping
5365 5366 5367 5368	Bell shape acc./ dec. time constant for rigid tapping (No. 5368 is used for the T series only.)
4037	Velocity loop feed-forward coefficient
4344	Advanced preview feed-forward coefficient (This parameter sets the coefficient for rapid traverse when bit 4 of parameter No. 4542 is 1.)
4542#4	Cutting feed/rapid traverse feed-forward function
4627	Advanced preview feed-forward coefficient for cutting feed (This parameter is valid when bit 4 of parameter No. 4542 is 1.)

Restriction

- If you use rough TTL resolution sensor such as α*i* Positioncoder(4096pulse/rev) in spindle position sensor, there is a possibility that the vibration or strange sound might be detected due to the following movement of servo axis to rough feedback data. We don't recommend to use such sensors for Smart rigid tapping. Use analog 1Vpp sensor or serial type sensor.

- You can't use Smart rigid tapping in the following conditions.
  - Using sensor-less motors as spindle motor
  - Using Spindle differential speed control

- Smart rigid tapping becomes invalid in the following kinds of rigid tappings. In these cases, Normal rigid tapping will be executed. When you use them, set some parameters such as time constant which are shown in "Parameter list related to Normal rigid tapping".
    - Peck rigid tapping
    - Rigid tapping by manual handling feed
    - Extraction in rigid tapping
    - Rigid tapping using Spindle control by servo motor
    - Rigid tapping by analog spindle

- In the situation that plural spindle axes are synchronized by Spindle command synchronous control mode, Smart rigid tapping becomes invalid. PS0203 alarm "PROGRAM MISS AT RIGID TAPPING" will occur.

- Smart rigid tapping becomes invalid in the checking mode in manual handle retrace. In the mode, Normal rigid tapping will be executed.

- As Smart rigid tapping executes acc./dec. by spindle maximum power, Acc./dec. after interpolation and Look-ahead acc./dec. before interpolation become invalid.

- In Smart rigid tapping over-ride signals for feed axes *FV0-*FV7<Gn012> and over-ride cancel signal OVC<Gn006.4> become invalid even if parameter OVS(No.5203#4)=1. Over-ride for extraction of rigid tapping is valid.

- In Smart rigid tapping the backlash compensation for spindle becomes invalid. The backlash compensation for feed axis becomes valid.

- Interlock and machine lock become invalid during smart rigid tapping. Axes become interlock state or machine lock state after finishing Smart rigid tapping.

- In Smart rigid tapping feed hold and single block become invalid. The axes don’t stop until the end of 1cycle of tapping.

- Reset and emergency stop during Smart rigid tapping make velocity command 0 stop.

- It’s impossible to change feedrate by external deceleration during Smart rigid tapping. Feedrate at starting cutting or extraction is maintained.

- The following functions become invalid during Smart rigid tapping.
    - General purpose retract
    - Tool retract and recover
    - One-digit F code feed
    - Manual handle interrupt

- When you use 3-dimensional rigid tapping, set D3R (Parameter No.11221#1) to “1”.

- The following settings are not supported for Smart rigid tapping axis.
    - Slave axis of superimposed control
    - Slave axis of flexible synchronization control
    - Slave axis of electronic gear box

- Alarm (SV0048) "SMART RIGID TAPPING STOP" will occur with the following alarms during Smart rigid tapping and the excitation of servo motor will be turned off. Alarm (SV0048) can be canceled with reset.
    - Alarm (SP9xxx) (when spindle motor is deactivated)

- Alarm (SP0741) (when synchronous error range exceeds the setting value of parameter (No.5214))

- When SRT(No.5206#1) is 1 and the spindle software, which doesn't support Smart rigid tapping, is used, alarm PS0564 "ILLEGAL COMMAND OF SMART RTAP" will occur at rigid tapping.

10.13.18 Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
2005							FEED	

[Input type] Parameter input
[Data type] Bit axis

#1 FEED Feed forward function is:
0: Disabled.
1: Enabled.

> **NOTE**
> Set 1 to the synchronizing servo axis with spindle axis on FSSB high-speed rigid tapping.
> In FSSB high-speed rigid tapping mode, the feed forward coefficient becomes 100 percent.

> **NOTE**
> This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in CONNECTION MANUAL (FUNCTION) (B-64693EN-1).

	#7	#6	#5	#4	#3	#2	#1	#0
2423						RGDBLA		

[Input type] Parameter input
[Data type] Bit axis

#2 RGDBLA Setting for the compensation of reverse operation in Smart rigid tapping
0: Compensation invalid
1: Compensation valid

	#7	#6	#5	#4	#3	#2	#1	#0
2429							FHRSV	

[Input type] Parameter input
[Data type] Bit axis

#1 FHRSV Servo axis control of FSSB high-speed rigid tapping is:
0: Disabled.
1: Enabled.

> **NOTE**
> Set 1 to the synchronizing servo axis of a spindle axis on FSSB high-speed rigid tapping.

2610	Position control loop gain for the tapping axis in FSSB high-speed rigid tapping

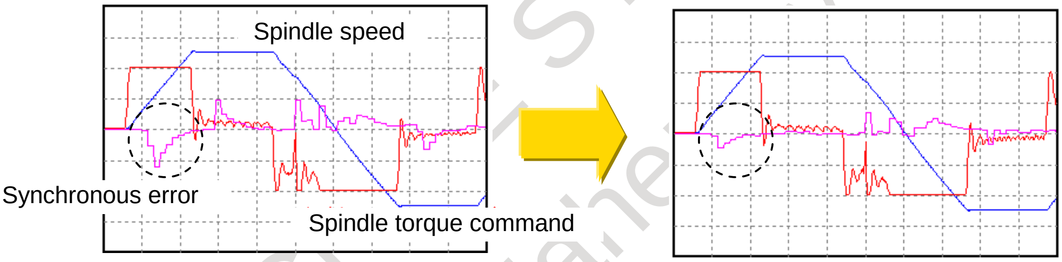
[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 32767

The position control loop gain for the tapping axis (servo axis) in FSSB high-speed rigid tapping can be set.
If the setting value of this parameter is 0, the position control loop gain is the same as the position gain (parameters Nos.5280 to 5284) of conventional rigid tapping used by both of servo axis and spindle axis.

2611	Coefficient of velocity feed-forward for feed axis in smart rigid tapping

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 1%
[Standard setting] 0

Use this parameter to improve the axis response in case that the synchronous error is large at the start point of tapping as follow figure. Increase this value to every 100 with watching the synchronous error. There is a possibility that the torque command of servo axis might become vibrating at acc./dec. if the value is too large.

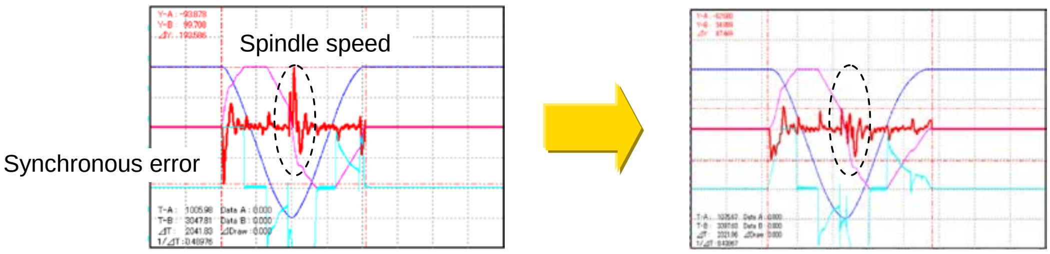


No.2611=0 No.2611=350

2613	Compensation of reverse operation in Smart rigid tapping

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01%
[Valid data range] 0 to 10000
[Standard setting] 0

Use this parameter to improve the axis response in case that the synchronous error becomes large at the reversal point of tapping. Please increase this value to every 100 with watching the synchronous error.



No.2613=0 No.2613=700

	#7	#6	#5	#4	#3	#2	#1	#0
3702							EMS	

[Input type] Parameter input
[Data type] Bit path

#1 EMS The multi-spindle control is:
0: Used.
1: Not used.

	#7	#6	#5	#4	#3	#2	#1	#0
3703				SPR	MPP	MPM		2P2

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 2P2 When a multi-path system is used, inter-path spindle control allows:
0: Configuration where the spindle that belongs to one path only is shared between path 1 and path 2.
1: Configuration where the spindles that belong to path 1 and 2 are shared between the two paths.
When the spindle that belongs to an arbitrary path is shared between arbitrary paths, set bit 2 (MPM) of parameter No. 3703. (The meanings of signals used vary, so that ladder program modifications need to be made.)

#2 MPM When a multi-path system is used, the configuration allowed by inter-path spindle control:
0: Follows the setting of bit 0 (2P2) of parameter No. 3703.
1: Allows the sharing of the spindle that belongs to a path between arbitrary paths.

#3 MPP In multi-spindle control, a spindle selection using a programmed command instead of using the signals (SWS1 to SWS4<Gn027.0 to 2, Gn026.3>) is:
0: Not made.
1: Made.

> **NOTE**
> When this parameter is set to 1, set parameter No. 3781 at the same time.

#4 SPR Rigid tapping with spindle of another path function is:
0: Not available.
1: Available.
The "rigid tapping with spindle of another path" function can be combined with the address P-based spindle select function. This combination enables program commands (P code) to be used to select a spindle in a path other than the command issuing path for rigid tapping.

	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS	SGT			ESF
		SFA	NSF		SGT			ESF

[Input type] Parameter input
[Data type] Bit path

#0 ESF When the spindle control function (Spindle analog output or Spindle serial output) is used, and the constant surface speed control function is used or bit 4 (GTT) of parameter No. 3706 is set to 1:

0: S codes and SF are output for all S commands.
1: For the lathe system:
 S codes and SF are not output for an S command in the constant surface speed control (G96) mode and a command for maximum spindle speed clamping (G92 S_; (G50 for G code system A)).
 For the machining center system:
 S codes and SF are not output for an S command in the constant surface speed control (G96) mode.

> **NOTE**
> The operation of this parameter varies between the lathe system and machining center system.
> For the lathe system:
> This parameter is valid when bit 4 (EVS) of parameter No. 3705 is set to 1.
> For the machining center system:
> For an S command for maximum spindle speed clamping (G92 S_;), SF is not output, regardless of the setting of this parameter.

#3 SGT Gear switching method during tapping cycle (G84 and G74) is:

0: Method A (Same as the normal gear switching method)
1: Method B (Gears are switched during tapping cycle according to the spindle speed set in parameters Nos. 3761 and 3762).

#4 EVS When the spindle control function (Spindle analog output or Spindle serial output) is used, S codes and SF are:

0: Not output for an S command.
1: Output for an S command.

The output of S codes and SF for an S command in constant surface speed control mode (G96), or for an S command used to specify maximum spindle speed clamping (G92 S_; (G50 for G code system A)) depends on the setting of bit 0 (ESF) of parameter No. 3705.

#5 NSF For the M series, when a T type gear is selected (with bit 4 (GTT) of parameter No. 3706 set to 1 or constant surface speed control is enabled (bit 0 (SSC) of parameter No. 8133 is 1)), and an S code is specified:

0: SF is output.
1: SF is not output.

> **NOTE**
> This parameter does not affect S code output. For an S command for maximum spindle speed clamping (G92 S_;), SF is not output, regardless of the setting of this parameter.

#6 SFA The SF signal is output:
0: When gears are switched.
1: Irrespective of whether gears are switched.

	#7	#6	#5	#4	#3	#2	#1	#0
3706								
				GTT				

[Input type] Parameter input
[Data type] Bit path

#4 GTT Spindle gear selection method is:
0: M type.
1: T type.

> **NOTE**
> 1 This parameter is effected for M system.
> 2 M type
> The gear selection signal is not input. The CNC selects a gear based on the speed range of each gear set by a parameter beforehand according to S codes, and the selected gear is posted by outputting the gear selection signal. Moreover, the spindle speed matching the gear selected by the output gear selection signal is output.
> T type
> The gear selection signal is input. The spindle speed matching the gear selected by this signal is output.
> 3 When the constant surface speed control is enabled (bit 0 (SSC) of parameter No. 8133 is 1), T type is selected, regardless of whether this parameter is specified.
> 4 When T type spindle gear switching is selected, the following parameters have no effect:
> No.3705#2(SGB), No.3751, No.3752,
> No.3705#1(GST), No.3705#3(SGT), No.3761, No.3762,
> No.3705#6(SFA), No.3735, No.3736
> On the other hand, parameter No. 3744 becomes usable.
> 5 When multi spindle control is used, select to T type.

3720	Number of position coder pulses

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

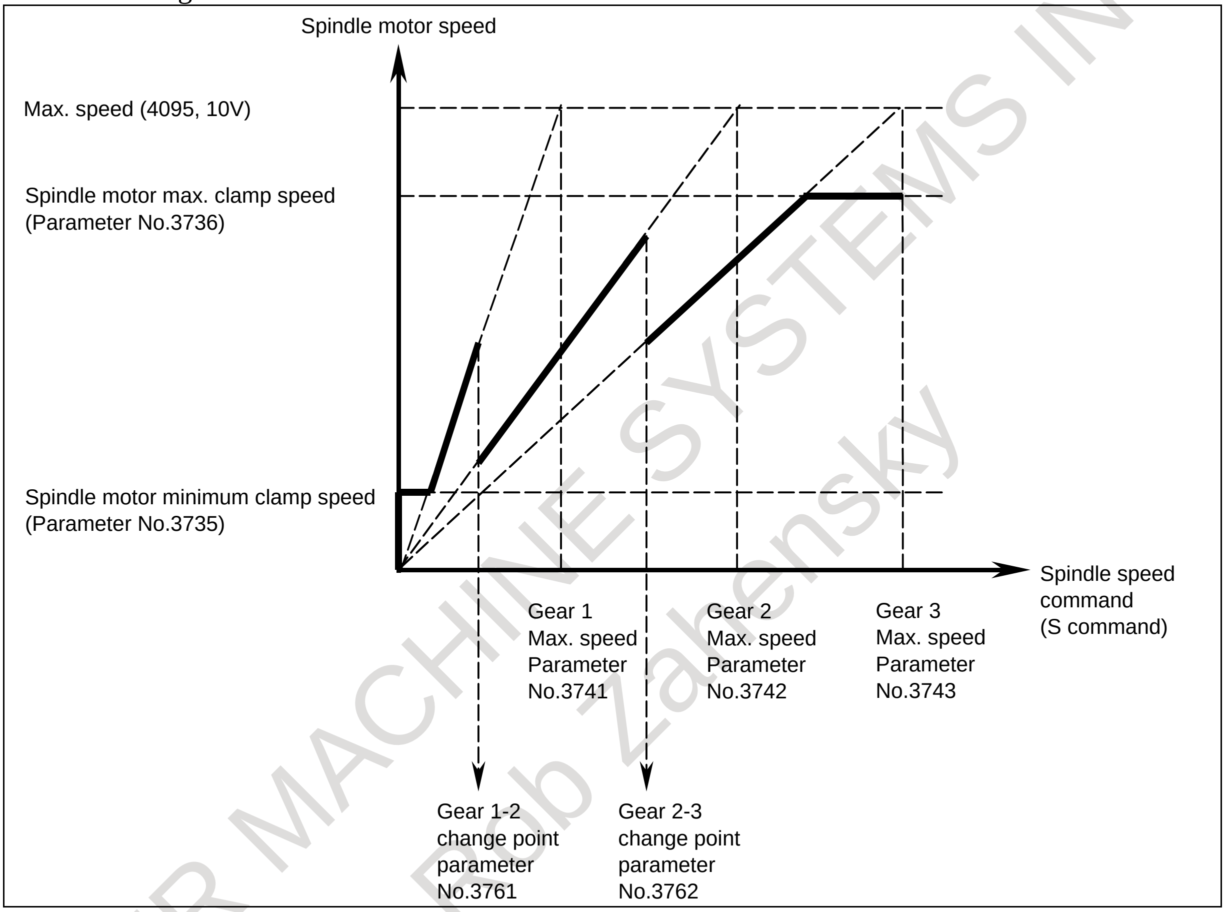
[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Detection unit
[Valid data range] 1 to 32767
Set the number of position coder pulses.

3761	
	Spindle speed when switching from gear 1 to gear 2 during tapping

3762	Spindle speed when switching from gear 2 to gear 3 during tapping

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

When method B is selected as the gear change method in the tapping cycle (when bit 3 (SGT) of parameter No. 3705 is set to 1), set the spindle speed at a change point of each gear.



	#7	#6	#5	#4	#3	#2	#1	#0
4000				RETSV				

[Input type] Parameter input
[Data type] Bit spindle

#4 RETSV Reference position return direction in the servo mode (rigid tapping, etc.)
0: The spindle returns to the reference position counterclockwise (CCW).
1: The spindle returns to the reference position clockwise (CW).

NOTE
This parameter is valid only when a serial spindle is used.

4073	Grid shift amount in the servo mode

[Input type] Parameter input
[Data type] Word spindle

[Unit of data] 1 pulse (360°/4096)
[Valid data range] 0 to 4095
Set this data when the reference position is to be shifted in the servo mode (rigid tapping, etc.).
When plus data is specified, the reference position for the spindle is shifted CCW by the set number of pulses.

> **NOTE**
> This parameter is valid only when a serial spindle is used.

4105	Time constant of magnetic flux for Magnetic flux control (for high-speed output switching characteristics)

4107	Time constant of magnetic flux for Magnetic flux control (for low-speed output switching characteristics)

[Input type] Parameter input
[Data type] Word spindle
The magnetic flux control is used to suppress the swelling of output power at acc./dec. in induction motors. These parameters are used for the function and the value is specific constant in each motor. When you use Smart rigid tapping, the setting of this parameter is necessary. If these parameters are 0 even after automatic parameter initialization, please ask FANUC of the setting value.

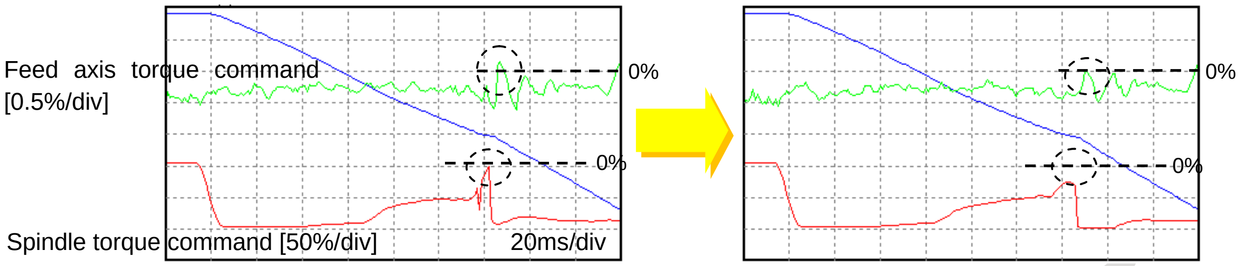
> **NOTE**
> When you use synchronous spindle motor (No.4012#6=1), the magnetic flux control is not necessary. Don’t set these parameters.

	#7	#6	#5	#4	#3	#2	#1	#0
4395			SRTDIF					

[Input type] Parameter input
[Data type] Bit spindle

#5 SRTDIF Improvement of deceleration for Smart rigid tapping is:
0: Invalid.
1: Valid.
These are some cases that shock sounds can be heard in the feed axis at the point where the spindle control changes from velocity control to position control during deceleration in tapping, if the torque command for spindle reverses and the torque command for feed axis reverses also as the result.
When this improvement becomes valid, the change from velocity control to positon control in spindle becomes smooth, and reverse of the spindle torque command is suppressed.
When you can’t suppress shock sounds even if this improvement is applied, use No.4790 together to set lower limit of torque command.

Spindle speed [500min^{-1}/div]



No.4395#5=0 No.4395#5=1

	#7	#6	#5	#4	#3	#2	#1	#0
4549							**FHRSP**	

[Input type] Parameter input
[Data type] Bit spindle

#1 FHRSP Spindle control of FSSB high-speed rigid tapping is:
0: Disabled.
1: Enabled.

> **NOTE**
> Set 1 to the synchronizing spindle axis with the servo axis on FSSB high-speed rigid tapping.

4661	**Tolerable error value at spindle stop**

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1/4096 revolution of spindle
[Valid data range] 0 to 2048
[Standard setting] 0

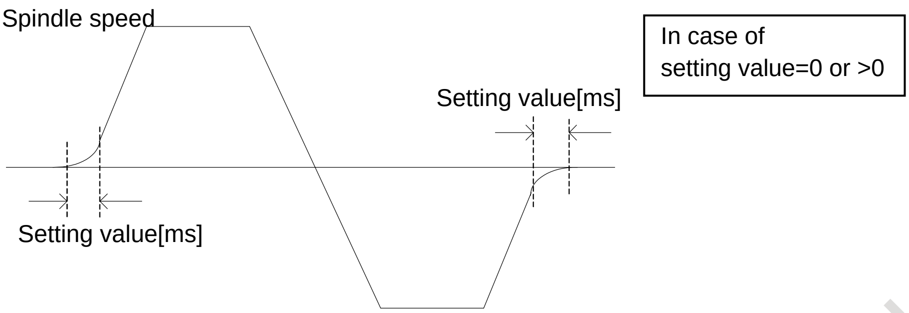
The tolerable error value at spindle stop should be set in No.4661. If the error at spindle stop becomes over this value, the alarm (SP9170) will occur.
The setting value 0 is treated as 1024. The standard setting value is 0.

4666	**Bell shape acc./dec. time constant for Smart rigid tapping CTH1A=0**

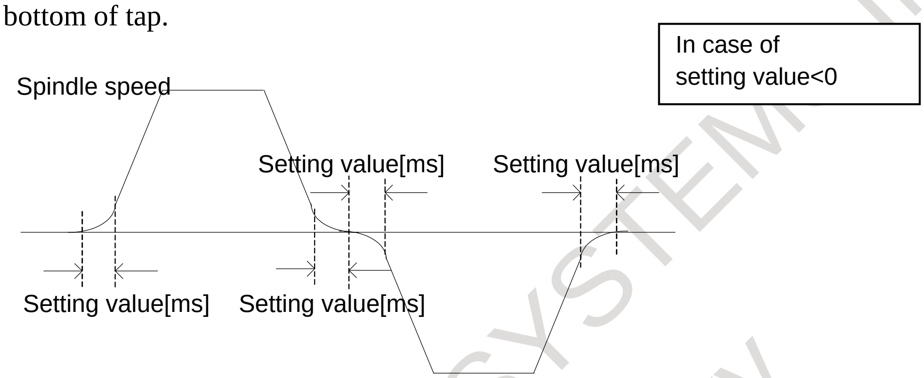
4667	**Bell shape acc./dec. time constant for Smart rigid tapping CTH1A=1**

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1ms
[Valid data range] -180 to 180
[Standard setting] 0

These parameters specify the bell shape acc./dec. time constant at starting from R point or ending point to R point. If there is a shock at acceleration or at deceleration in servo or spindle axis, please tune these parameters. The setting value 0 is treated as 30 (30ms).



When the setting value is negative, the bell shape acc./dec. becomes effective also at bottom of tap.



4668	Upper limit of torque command for Smart rigid tapping CTH1A=0

4669	Upper limit of torque command for Smart rigid tapping CTH1A=1

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%
[Valid data range] 0 to 100
[Standard setting] 0

These parameters specify the upper limit of torque command for Smart rigid tapping. If the following limits have been already applied, they are valid also in Smart rigid tapping.

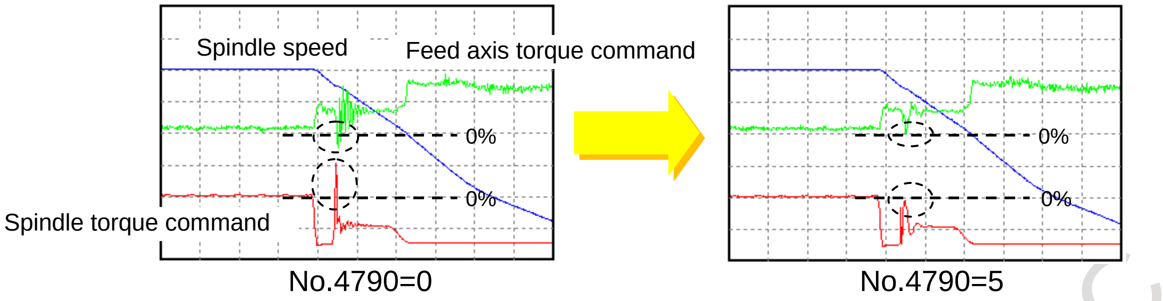
- Torque limit by parameter No.4025 and DI signal TLMH,L
- Power limit by parameter No.4028 and 4029

In such cases, it's not necessary to set these parameters. The setting value 0 is treated as 100 (100%).

4790	Lower limit of torque command for Smart rigid tapping in position control

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%
[Valid data range] 0 to 25
[Standard setting] 0

This parameter specifies the lower limit of torque command for spindle in position control and it clamps the torque command so that the torque command for spindle doesn't reverse. When you don't want the spindle torque command becomes under n%, Set n in this parameter.
If you set 0 in this parameter, the clamp of torque command is not done.



	#7	#6	#5	#4	#3	#2	#1	#0
5101								FXY

[Input type] Parameter input
[Data type] Bit path

#0 FXY The drilling axis in the drilling canned cycle, or cutting axis in the grinding canned cycle is:
0: Z-axis at all times.
1: Axis selected by the program

> **NOTE**
> In the case of the lathe system, this parameter is valid only for the drilling canned cycle in the FANUC Series 15 program format.

	#7	#6	#5	#4	#3	#2	#1	#0
5200	SRS	FHD	PCP	DOV	SIG	CRG		G84
		FHD	PCP	DOV	SIG	CRG		G84

[Input type] Parameter input
[Data type] Bit path

#0 G84 Method for specifying rigid tapping:
0: An M code specifying the rigid tapping mode is specified prior to the issue of the G84 (or G74) command. (See parameter No. 5210).
1: An M code specifying the rigid tapping mode is not used. (G84 cannot be used as a G code for the tapping cycle; G74 cannot be used for the reverse tapping cycle.)

#2 CRG Rigid mode when a rigid mode cancel command is specified (G80, 01 group G code, reset, etc.) :
0: Canceled after rigid tapping signal RGTAP is set to 0.
1: Canceled before rigid tapping signal RGTAP is set to 0.

#3 SIG When gears are changed for rigid tapping, the use of SINDs is:
0: Not permitted.
1: Permitted.

#4 DOV Override during extraction in rigid tapping:
0: Invalidated
1: Validated (The override value is set in parameter No. 5211. However, set an override value for retraction for rigid tapping in parameter No. 5381.)

#5 PCP Rigid tapping:
0: Used as a high speed peck tapping cycle
1: Not used as a high speed peck tapping cycle

#6 FHD Feed hold and single block in rigid tapping:
0: Invalidated
1: Validated

#7 SRS To select a spindle used for rigid tapping in multi-spindle control:
0: The spindle selection signals SWS1, SWS2, SWS3, and SWS4 are used. (These signals are used also for multi-spindle control.)
1: The rigid tapping spindle selection signals RGTSP1, RGTSP2, RGTSP3, and RGTSP4 are used. (These signals are provided expressly for rigid tapping.)

	#7	#6	#5	#4	#3	#2	#1	#0
5201				OV3	OVU	TDR		

[Input type] Parameter input
[Data type] Bit path

#2 TDR Cutting time constant in rigid tapping:
0: Uses a same parameter during cutting and extraction (Parameters Nos. 5261 to 5264)
1: Not use a same parameter during cutting and extraction
Parameters Nos. 5261 to 5264: Time constant during cutting
Parameters Nos. 5271 to 5274: Time constant during extraction

#3 OVU The increment unit of the override parameter No. 5211 for tool rigid tapping extraction is:
0: 1%
1: 10%

#4 OV3 A spindle speed for extraction is programmed, so override for extraction operation is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5202				IRR				ORI

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 ORI When rigid tapping is started:
0: Spindle orientation is not performed.
1: Spindle orientation is performed.
This parameter can be used only for a serial spindle.
This spindle orientation performs reference position return in the serial spindle/servo mode. The stop position can be changed using the serial spindle parameter No. 4073.

#4 IRR As the in-position width at point R after movement from point I to point R in rigid tapping:
0: The in-position widths dedicated to rigid tapping (parameters Nos. 5300, 5302, 5304, and 5306) are selected.
1: The normal in-position width (parameter No. 1826) is selected.

	#7	#6	#5	#4	#3	#2	#1	#0
5203			RBL	OVS		RFF		

[Input type] Parameter input
[Data type] Bit path

#2 RFF In rigid tapping, feed forward is:
0: Disabled.
1: Enabled. (Recommended)

As the standard setting, set 1.
At the same time, set the parameter for the advanced preview feed forward coefficient for the tapping axis and the parameter for the advance preview feed forward coefficient for the spindle so that these values match.

- Advanced preview feed forward coefficient for the tapping axis: Parameter No. 2092 (or parameter No. 2144 if the cutting/rapid traverse feed forward function is enabled (bit 4 (FFCHG) of parameter No. 2214 is set to 1))
- Advanced preview feed forward coefficient for the spindle: Parameter No. 4344

> **NOTE**
> This parameter is valid when a serial spindle is used.

> **NOTE**
> This parameter is automatic setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in CONNECTION MANUAL (FUNCTION) (B-64693EN-1).

#4 OVS In rigid tapping, override by the feedrate override select signal and cancellation of override by the override cancel signal is:
0: Disabled.
1: Enabled.
When feedrate override is enabled, extraction override is disabled.
The spindle override is clamped to 100% during rigid tapping, regardless of the setting of this parameter.

#5 RBL As acceleration/deceleration for rigid tapping cutting feed:
0: Linear acceleration/deceleration is used.
1: Bell-shaped acceleration/deceleration is used.

> **NOTE**
> This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in CONNECTION MANUAL (FUNCTION) (B-64693EN-1).

	#7	#6	#5	#4	#3	#2	#1	#0
5206	RSP					NSD	SRT	

[Input type] Parameter input
[Data type] Bit

#1 SRT Smart rigid tapping function bit
0: Function invalid
1: Function valid

#2 NSD Rigid tapping during dry run is:
- 0: Smart rigid tapping (Change of manual feedrate override signal *JV0 to *JV15<Gn010, Gn011> or manual rapid traverse selection signal RT<Gn019.7> is not effective.).
- 1: FSSB high-speed rigid tapping (Change of manual feedrate override signal *JV0 to *JV15<Gn010, Gn011> or manual rapid traverse selection signal RT<Gn019.7> is effective.).

#7 RSP Rigid tapping synchronization error range is:
- 0: After reaching point R and the changing of the loop gain and parameter of position control being completed, the calculation of rigid tapping synchronization error starts with the movement of tapping axis.
- 1: After reaching point R and the changing of the loop gain and parameter of position control being completed, the calculation of rigid tapping synchronization error starts immediately.

	#7	#6	#5	#4	#3	#2	#1	#0
5209	PRA	CSA				DWP	RIP	RTX
	PRA	CSA					RIP	

[Input type] Parameter input
[Data type] Bit path

#0 RTX In rigid tapping in a lathe system, the tapping axis is:
- 0: Selected by selecting a plane.
- 1: Always assumed to be the Z-axis for G84 or the X-axis for G88.

> **NOTE**
> This parameter becomes invalid when bit 1 (FCV) of parameter No. 0001 is set to 1, and rigid tapping is specified using the FANUC Series 15 program format.

#1 RIP When a movement from the initial point to point R is made, the in-position check is:
- 0: Dependent on the setting of bit 5 (NCI) of parameter No. 1601.
- 1: Performed.

> **NOTE**
> This parameter is valid when bit 5 (NCI) of parameter No. 1601 is set to 1 and bit 4 (IRR) of parameter No. 5202 is set to 0.
> If bit 5 (NCI) of parameter No. 1601 is set to 0, the in-position check is performed regardless of the setting of this parameter.

#2 DWP When a dwell (address P) command is not included in a block for lathe-system rigid tapping:
- 0: Dwelling at the bottom of a hole is not performed.
- 1: The dwell (address P) command specified in the block for drilling is valid.

> **NOTE**
> This parameter becomes invalid if rigid tapping is specified in the FANUC Series 15 program format with bit 1 (FCV) of parameter No. 0001 set to 1.

#6 CSA If Constant surface speed control is commanded in Rigid tapping mode:
0: Alarm is not issued.
1: Alarm PS0200, "ILLEGAL S CODE COMMAND" is issued.

> **NOTE**
> This parameter is automatic setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in CONNECTION MANUAL (FUNCTION) (B-64693EN-1).

#7 PRA On the peck rigid tapping, if the depth of cut (Q) is smaller than the cutting start distance (d):
0: Alarm is not issued.
1: Alarm PS5560, "ILLEGAL DEPTH OF CUT" is issued.

> **NOTE**
> 1 When High-speed peck rigid tapping cycle is not used (bit 5 (PCP) of parameter No.5200 is 1), this parameter is effective.
> 2 When FSSB high-speed rigid tapping is used, set 1 to this parameter.

> **NOTE**
> This parameter is automatic setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in CONNECTION MANUAL (FUNCTION) (B-64693EN-1).

5210	Rigid tapping mode specification M code

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 65535
This parameter sets an M code that specifies the rigid tapping mode.
The M code is judged to be 29 (M29) when 0 is set.

5211	Override value during rigid tapping extraction

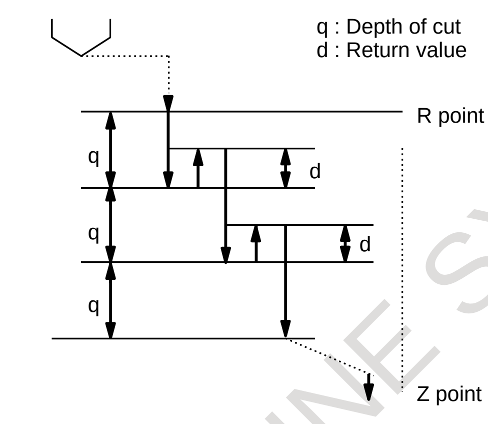
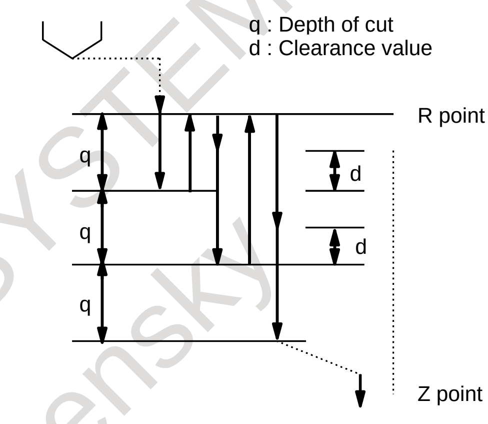
[Input type] Parameter input
[Data type] Word path
[Unit of data] 1% or 10%
[Valid data range] 0 to 200
The parameter sets the override value during rigid tapping extraction.

> **NOTE**
> 1 The override value is valid when bit 4 (DOV) of parameter No. 5200 is set to 1. When bit 3 (OVU) of parameter No. 5201 is set to 1, the unit of set data is 10% and an override of up to 2000% can be applied to extraction.
> 2 If the setting value is less than or equal to 0, it is treated as 100%. Also, if the setting value is greater than 200, it is treated as 2000% or 200%.

5213	Return in peck rigid tapping cycle

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the drilling axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the return or clearance in the peck tapping cycle.
If the setting value is negative value, it is treated as absolute value.

When the bit 5 (PCP) of parameter No. 5200 is set to 0. (High speed peck tapping cycle)	When the bit 5 (PCP) of parameter No. 5200 is set to 1. (Peck tapping cycle)
	

> **NOTE**
> CNC reads a setting value of parameter No.5213 when a block of peck rigid tapping cycle is read (buffered). If a setting value of parameter No.5213 is changed during automatic operation, a setting value has to be changed when buffering is prevented by M code preventing buffering and so forth.

5214	Setting of an allowable rigid tapping synchronization error range

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

This parameter sets an allowable synchronization error range in rigid tapping.
If a synchronous error range exceeding the setting of this parameter is detected, the alarm SP0741, “RIGID TAP ALARM : EXCESS ERROR” is issued. When 0 is set in this parameter, no synchronization error check is made.

<u>For Smart rigid tapping</u>
Synchronization error check is always executed in Smart rigid tapping. In regard to spindle using Smart rigid tapping, the setting value 0 is treated as 400. The standard setting value is 0, but change it if you need.

5221	Number of gear teeth on the spindle side in rigid tapping (first gear)
5222	Number of gear teeth on the spindle side in rigid tapping (second gear)
5223	Number of gear teeth on the spindle side in rigid tapping (third gear)
5224	Number of gear teeth on the spindle side in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 1 to 32767

Each of these parameters is used to set the number of gear teeth on the spindle side for each gear in rigid tapping.

> **NOTE**
> When a position coder is attached to the spindle, set the same value for all of parameters Nos. 5221 to 5224.

5231	Number of gear teeth on the position coder side in rigid tapping (first gear)
5232	Number of gear teeth on the position coder side in rigid tapping (second gear)
5233	Number of gear teeth on the position coder side in rigid tapping (third gear)
5234	Number of gear teeth on the position coder side in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 1 to 32767

Each of these parameters is used to set the number of gear teeth on the position coder side for each gear in rigid tapping.

> **NOTE**
> When a position coder is attached to the spindle, set the same value for all of parameters Nos. 5231 to 5234.

5241	Maximum spindle speed in rigid tapping (first gear)
5242	Maximum spindle speed in rigid tapping (second gear)
5243	Maximum spindle speed in rigid tapping (third gear)
5244	Maximum spindle speed in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 9999

Spindle position coder gear ratio
1 : 1 0 to 7400
1 : 2 0 to 9999
1 : 4 0 to 9999
1 : 8 0 to 9999

Each of these parameters is used to set a maximum spindle speed for each gear in rigid tapping.

Set the same value for both parameters Nos. 5241 and 5243 for a one-stage gear system. For a two-stage gear system, set the same value as set in parameter No. 5242 in parameter No. 5243. Otherwise, alarm PS0200, "ILLEGAL S CODE COMMAND" will be issued. This applies to the machining center system.

5261	Time constant for acceleration/deceleration in rigid tapping for each gear (first gear)
5262	Time constant for acceleration/deceleration in rigid tapping for each gear (second gear)
5263	Time constant for acceleration/deceleration in rigid tapping for each gear (third gear)
5264	Time constant for acceleration/deceleration in rigid tapping for each gear (fourth gear)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] msec
[Valid data range] 0 to 4000

Each of these parameters is used to set a linear acceleration/ deceleration time constant for the spindle of each gear and the tapping axis in rigid tapping.
Set the period required to reach each maximum spindle speed (parameters Nos. 5241 to 5244).
The set time constant, multiplied by the ratio of a specified S value to a maximum spindle speed, is actually used as a time constant.
For bell-shaped acceleration/deceleration, set a time constant for a linear portion.

5271	Time constant for acceleration/deceleration in rigid tapping extraction (first gear)
5272	Time constant for acceleration/deceleration in rigid tapping extraction (second gear)
5273	Time constant for acceleration/deceleration in rigid tapping extraction (third gear)
5274	Time constant for acceleration/deceleration in rigid tapping extraction (fourth gear)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] msec
[Valid data range] 0 to 4000

Each of these parameters is used to set a linear acceleration/ deceleration time constant for the spindle of each gear and tapping axis in extraction operation during rigid tapping.
For bell-shaped acceleration/deceleration, set a time constant for a linear portion.

NOTE

These parameters are enabled when the parameter TDR (bit 2 of parameter No. 5201) is set to 1.

5280	Position control loop gain for the spindle and tapping axis in rigid tapping (common to gears)
5281	Position control loop gain for the spindle and tapping axis in rigid tapping (first gear)
5282	Position control loop gain for the spindle and tapping axis in rigid tapping (second gear)
5283	Position control loop gain for the spindle and tapping axis in rigid tapping (third gear)

5284	Position control loop gain for the spindle and tapping axis in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 0.01/sec
[Valid data range] 1 to 9999

Each of these parameters is used to set a position control loop gain for the spindle and tapping axis in rigid tapping. These parameters significantly affect the precision of threading. Optimize these parameters as well as the loop gain multipliers by conducting a cutting test.

NOTE
To use a varied loop gain on a gear-by-gear basis, set parameter No. 5280 to 0, and set a loop gain for each gear in parameters Nos. 5281 to 5284. The specification of a loop gain on a gear-by-gear basis is disabled if parameter No. 5280 is set to a value other than 0. In such a case, the value set in parameter No. 5280 is used as a loop gain that is common to all the gears.

5291	Loop gain multiplier for the spindle in rigid tapping (first gear)
5292	Loop gain multiplier for the spindle in rigid tapping (second gear)
5293	Loop gain multiplier for the spindle in rigid tapping (third gear)
5294	Loop gain multiplier for the spindle in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 1 to 32767

Each of these parameters is used to set a loop gain multiplier for the spindle in rigid tapping each gear.
These parameters significantly affect the precision of threading. Optimize these parameters as well as the loop gains by conducting a cutting test.
Loop gain multiplier GC is obtained from the following equation:

$$GC = \frac{2048000 \times 360 \times PC \times E}{PLS \times SP \times L}$$

PLS Number of pulses output from the position coder (pulses/rev)
SP Number of gear teeth on the spindle side
PC Number of gear teeth on the position coder side
E Specified voltage (V) for turning the spindle motor at 1000 min^{-1}
L Angular displacement of the spindle (degrees) per spindle motor rotation

[Example] For the spindle motor and gear ratio given below, GC is calculated as follows:

$$GC = \frac{2048000 \times 360 \times 1 \times 2.2}{4096 \times 1 \times 360} = 1100$$

PLS = 4096 pulse/rev
SP = 1
PC = 1
E = 2.2 V
L = 360 deg

(Note) On the assumption that the spindle motor used turns at 4500 min^{-1} at 10 V, 2.2 V is required to turn the spindle motor at 1000 min^{-1}.

> **NOTE**
> These parameters are used for analog spindles.

5300	Tapping axis in-position width in rigid tapping (first spindle)

5302	Tapping axis in-position width in rigid tapping (second spindle)

5304	Tapping axis in-position width in rigid tapping (third spindle)

5306	Tapping axis in-position width in rigid tapping (fourth spindle)

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] 0 to 32767

This parameter sets a tapping axis in-position width when rigid tapping is performed using the each spindle.
If the setting value is negative value, in-position check is not completed.

5301	Spindle in-position width in rigid tapping

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 32767

This parameter are used to set spindle in-position width in rigid tapping.
If the setting value is negative value, in-position check is not completed.

> **NOTE**
> If an excessively large value is specified, the threading precision will deteriorate.

5310	Positional deviation limit imposed during tapping axis movement in rigid tapping (first spindle)

5350	Positional deviation limit imposed during tapping axis movement in rigid tapping (second spindle)

5354	Positional deviation limit imposed during tapping axis movement in rigid tapping (third spindle)

5358	Positional deviation limit imposed during tapping axis movement in rigid tapping (fourth spindle)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

This parameter sets a positional deviation limit imposed during tapping axis movement in rigid tapping using the each spindle.

5311	Limit value of spindle positioning deviation during movement in rigid tapping

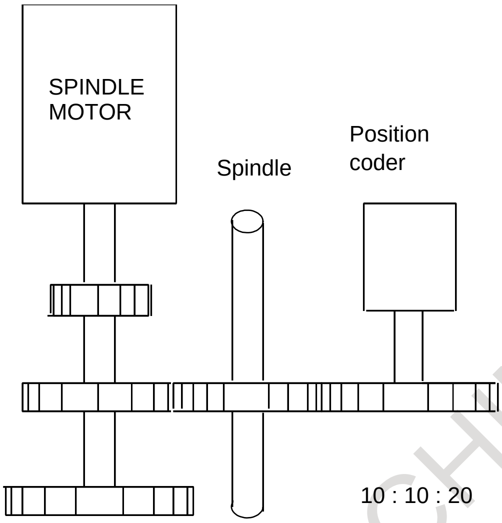
[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

This parameter sets the limit value of a spindle positioning deviation during movement in rigid tapping.
Find a value to be set from the following expression:

$$Setting\ value = \frac{S \times PLS \times 100 \times SP \times C}{60 \times G \times PC}$$

S Maximum spindle speed in rigid tapping (min^{-1})
(Setting value of parameters Nos. 5241 and to 5244)
PLS Number of pulses output from the position coder (pulses/rev)
SP Number of gear teeth on the spindle side
PC Number of gear teeth on the position coder side
G Loop gain in the rigid tapping ($0.01sec^{-1}$)
(Setting value of parameters Nos. 5281 and to 5284)
C Coefficient 1.5

[Calculation example]



S = 3600
PLS = 4096
SP = 10
PC = 20
G = 3000
C = 1.5

$$Setting\ value = \frac{3600 \times 4096 \times 100 \times 10 \times 1.5}{60 \times 3000 \times 20} = 6144$$

5312	**Positional deviation limit imposed while the tapping axis is stopped in rigid tapping (first spindle)**
5352	**Positional deviation limit imposed while the tapping axis is stopped in rigid tapping (second spindle)**
5356	**Positional deviation limit imposed while the tapping axis is stopped in rigid tapping (third spindle)**
5360	**Positional deviation limit imposed while the tapping axis is stopped in rigid tapping (fourth spindle)**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] 0 to 32767

This parameter sets a positional deviation limit imposed while the tapping axis is stopped in rigid tapping using the each spindle.

5313	**Positional deviation limit imposed while the spindle is stopped in rigid tapping**

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

This parameter is used to set a positional deviation limit imposed while the spindle is stopped in rigid tapping.

5321	Spindle backlash in rigid tapping (first-stage gear)
5322	Spindle backlash in rigid tapping (second-stage gear)
5323	Spindle backlash in rigid tapping (third-stage gear)
5324	Spindle backlash in rigid tapping (fourth-stage gear)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit ([detection unit]=[spindle rotation angle per one rotation of position coder]/4096 for αi position coder)
[Valid data range] -9999 to 9999
Each of these parameters is used to set a spindle backlash in rigid tapping.

5365	Bell-shaped acceleration/deceleration time constant in rigid tapping (first-stage gear)
5366	Bell-shaped acceleration/deceleration time constant in rigid tapping (second-stage gear)
5367	Bell-shaped acceleration/deceleration time constant in rigid tapping (third-stage gear)
5368	Bell-shaped acceleration/deceleration time constant in rigid tapping (fourth-stage gear)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] msec
[Valid data range] 0 to 512
Each of these parameters is used to set a time constant for a curved portion when bell-shaped acceleration/deceleration is selected in rigid tapping. When 0 is set in this parameter, linear acceleration/ deceleration is performed.

NOTE
This parameter is enabled when the bit 5 (RBL) of parameter No. 5203 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
24203								FHR

NOTE
When the parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 FHR Position data transmission by FSSB is:
0: Disabled.
1: Enabled.

NOTE
1 In addition, it is necessary to set parameter No.24204.
2 When FSSB high-speed rigid tapping is used, set 1 to this parameter.

24204	The index number of the spindle axis that synchronizes to each servo axis

> **NOTE**
> When the parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to the maximum number of spindles

In FSSB high-speed rigid tapping, set the index number of the spindle axis that synchronizes to servo axis.

> **NOTE**
> 1 When the bit 0 (FHR) of parameter No.24203 is 1, this parameter is enabled.
> 2 Eight or less index numbers of the spindle axis can be set in the system. If nine or more index numbers of the spindle axis are set, the alarm PW0037, "SV/SP COMBINATION ERROR" is issued.

24208	The index number of the spindle axis that synchronizes to each servo axis (2nd set)

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to the maximum number of spindles

This parameter sets the index number of the spindle synchronizing with the servo axis used in FSSB high-speed rigid tapping. Whether to use parameter No.24204 or this parameter can be changed by Spindle/Servo axis combination change signal SPSVS<G0518.6>.

> **NOTE**
> 1 This parameter is effective when parameter FHR (No.24203#0)=1.
> 2 Eight or less index numbers of the spindle axis can be set in the system. If nine or more are set, the alarm (DS0092) "SV/SP COMBINATION ERROR" is issued.

Alarm and message

Number	Message	Description
PS0200	ILLEGAL S CODE COMMAND	(1) In the rigid tap, an S value was out of range or was not specified. The parameters Nos. 5241 to 5243 setting is an S value which can be specified for the rigid tapping. Correct the parameters or modify the program. (2) Rigid tapping is commanded during constant surface speed control. Command rigid tapping after canceling constant surface speed control.
PS0201	FEEDRATE NOT FOUND IN RIGID TAP	The command F code for a cutting feedrate is a zero. If the value of F command is much smaller than that of the S command, when a rigid tapping command is specified, this alarm is generated. This is because cutting is not possible by the lead specified by the program.
PS0202	POSITION LSI OVERFLOW	In the rigid tap, spindle distribution value is too large. Decrease spindle speed.

Number	Message	Description
PS0203	PROGRAM MISS AT RIGID TAPPING	(1) In the rigid tap, position for a rigid M code (M29) or an S command is incorrect. Modify the program. (2) FSSB high-speed rigid tapping cannot be used on the rigid tapping with one servo motor and multiple spindle motors by spindle command synchronous control. (3) In FSSB high-speed rigid tapping, parameter setting is wrong (bits 1 and 2 of parameter No.2429, and bits 1 and 2 of parameter No.4549). (4) In FSSB high-speed rigid tapping, parameter No.24204 is wrong. Set the same index number of the spindle synchronizing to three axes of 3-dimensional rigid tapping.
PS0204	ILLEGAL AXIS OPERATION	In the rigid tap, an axis movement is specified between the rigid M code (M29) block and G84 (or G74) block. Modify the program.
PS0205	RIGID MODE DI SIGNAL OFF	(1) Although a rigid M code (M29) is specified in rigid tapping, the rigid mode DI signal (Gn061.0) is not ON during execution of the G84 (or G74) block. Check the PMC ladder diagram to find the reason why the DI signal is not turned on. (2) In rigid tapping, no spindle has selected commands in the path of interest. Alternatively, spindles in two or more paths have selected commands in the same path. Check the spindle command select signal. (3) A spindle specified for rigid tapping is already being used in another path. (4) In a machine provided with the multi-spindle control, no spindle has been selected for attempted rigid tapping. (5) Rigid tapping command is issued to a spindle control axis with servo motor in a path other than the command issuing path.
PS0206	CAN NOT CHANGE PLANE (RIGID TAP)	Plane changeover was instructed in the rigid mode. Modify the program.
PS0207	RIGID DATA MISMATCH	The specified distance was too short or too long in rigid tapping.
PS0564	ILLEGAL COMMAND OF SMART RTAP	The command for Smart rigid tapping is illegal. The reason of this alarm is in the followings. - The moving distance of feed axis is larger than 4096 times of screw pitch. - The used spindle software doesn't support Smart rigid tapping.
PS5560	ILLEGAL DEPTH OF CUT	On the peck rigid tapping, the depth of cut (Q) is smaller than the cutting start distance (d).
SV0048	SMART RIGID TAPPING STOP	The excitation of servo motors is turned off because smart rigid tapping is stopped. This alarm is accompanied by spindle alarm.
SV0410	EXCESS ERROR (STOP)	The amount of positional deviation during stopping exceeded the parameter No. 1829 setting value.
SV0411	EXCESS ERROR (MOVING)	The amount of positional deviation during traveling became excessive than the parameter setting value.
SV0417	ILL DGTL SERVO PARAMETER	A digital serve parameter setting is incorrect.
SP0740	RIGID TAP ALARM : EXCESS ERROR	The positional deviation of the stopped spindle has exceeded the set value of parameter No.5313 during rigid tapping.

Number	Message	Description
SP0741	RIGID TAP ALARM : EXCESS ERROR	(1) The positional deviation of the moving spindle has exceeded the set value of parameter No.5311 during rigid tapping. (2) Rigid tapping synchronization error has exceeded the set value of parameter No.5214.
SP0742	RIGID TAP ALARM : LSI OVERFLOW	An LSI overflow has occurred for the spindle side during rigid tapping.
SP0752	SPINDLE MODE CHANGE ERROR	The spindle is in a mode (such as Cs contour control or spindle synchronous control) other than the spindle control mode. When the "rigid tapping with spindle of another path" function is used, the spindle to be used for rigid tapping must be switched to the spindle control mode before the start of rigid tapping.
PW0037	SV/SP COMBINATION ERROR	(1) The servo axis might be a setting of the dummy axis. Check to see the parameters No.1023, bit 0 of No.2009, and bit 4 of No.11802. (2) The index number of spindle axis that synchronizes servo axis is wrong. Check to see the parameters bit 0 of No.3716, No.3717, and No.24204. (3) FSSB high-speed rigid tapping cannot be used with the analog spindle or the spindle control with servo motor.
DS0092	SV/SP COMBINATION ERROR	1. The servo axis might be a setting of the dummy axis. Check to see the parameters No.1023, bit 0 of No.2009, and bit 4 of No.11802. 2. The index number of spindle axis that synchronizes servo axis is wrong.. Check parameters (bit 0 of No.3716, No.3717, No.24208). 3. One of the following functions is applied. Turn it off. - Rigid tapping - Servo/Spindle synchronous control - Electronic gear box

10.13.19 Notes

- Notes on spindles

Note

NOTE

1 A spindle pitch error is not compensated for in rigid tapping mode.

2 The limits to the number of pulses assigned to each spindle are as follows (displayed with diagnosis data No. 0451):
- Serial spindle: 32767 pulses every 8 msec
- Analog spindle: 4095 pulses every 8 msec

These values vary with the position coder gear ratio setting and rigid tapping specification. If a value greater than the maximum allowable number is specified, an alarm PS0202 is issued.

3 If rigid tapping is commanded during constant surface speed control, alarm PS0200, "ILLEGAL S CODE COMMAND" is issued. Command rigid tapping after canceling constant surface speed control.

- Other cautions

⚠ CAUTION

1 When the spindle orientation function is to be used at the same time
The spindle orientation function positions the spindle by using sensors and the PMC, without being directly controlled by the CNC.
The CNC has no direct control over this processing, instead following the specifications of the spindle orientation function being used.

2 When the spindle positioning function is to be used at the same time
When the spindle positioning function is to be used together with rigid tapping, rigid tapping mode must not be specified in spindle indexing mode, and spindle indexing mode must not be specified in rigid tapping mode. (Spindle positioning and rigid tapping cannot be performed simultaneously for a single spindle.)
This is not applicable when multi-spindle control is used together and a spindle used for spindle position indexing is different from a spindle used for rigid tapping.

⚠ **CAUTION**

3 When the Cs contouring control function is to be used at the same time
When the Cs contouring control function for the serial spindle is used together with the rigid tapping function, the same motor is used for spindle rotation control, Cs contouring control, and rigid tapping modes. The following points must be noted:

(1) Whether to enter Cs contouring control mode or spindle rotation control mode is selected by the CON (Cs contouring control switch signal) signal; however, the system can enter rigid tapping mode regardless of the state of the CON signal. When the rigid tapping mode is canceled the system enters spindle rotation control mode or Cs contouring control mode according to the state of the CON signal.

(2) Since the system can change to rigid tapping mode directly from the Cs contouring control mode, use of the Cs contouring control function enables the tapping tool to be positioned before rigid tapping begins. Accurate positioning is not guaranteed. As a matter of course, if the rigid tapping cycle executes gear change or output range changing, it becomes invalid when losing the position of the spindle..

(3) Although the system can change to rigid tapping mode directly from Cs contouring control mode, positions designated in Cs contouring control mode are not preserved if rigid tapping mode is canceled by G80 etc. When the system is changed to rigid tapping mode directly from Cs contouring control mode and wanting to set Cs contouring control mode again, then returns to the Cs contouring control mode, G00 or G28 must be issued to return to the reference position.

(4) In systems with the serial spindle Cs contouring control function, the spindle motor is in a state called servo mode when it is operating in rigid tapping mode. In servo mode, it can accept jog feed and manual handle feed. To prevent this, inhibit jog feed and manual handle feed of the Cs contouring axis in the PMC logic during rigid tapping.

(5) The servo-off signal for the Cs contour control axis should be masked on the PMC side during rigid tapping. It should be masked on the PMC side as required.

(6) When multi-spindle control is used together and a spindle used for Cs contour control is different from a spindle used for rigid tapping, rigid tapping can be specified with one spindle while the other spindle is placed in the Cs contour control mode.

(7) The Cs contour control axis cannot be a drilling axis. If it is specified as a drilling axis, alarm PS0203 is issued.

4 Program restart
The program cannot be restarted in a block placed in the rigid tapping mode. (The program cannot be restarted in blocks between an M29 command and G80 command.)

5 Specification of multiple M codes in one block
Specify an M code for specifying the rigid tapping mode as an independent M code. (Multiple M codes must not be specified in one block.)

6 Positioning by optimum accelerations
Positioning by optimum accelerations is disabled during rigid tapping.

- Position control loop gain switching and serial spindle parameters

In rigid tapping, the loop gain of the drilling axis is switched so that the loop gains for position control of the drilling axis and spindle match each other.
This switching processing is specified by parameters Nos. 5281 to 5284.
The spindle loop gain depends on the setting of serial spindle parameters and the input gear signals (CTH2, CTH1).
So, in order to perform rigid tapping with a serial spindle, the position control loop gain of the spindle must be set in the parameters for the serial spindle used for rigid tapping.
When multi-spindle control is used, a spindle other than the first spindle can be used for rigid tapping. So, set the parameters below for the serial spindle used for rigid tapping.
The parameters indicated below are the major serial spindle parameters required for the setting and adjustment needed to use a serial spindle.
For details of the serial spindle parameters, refer to the "FANUC AC SPINDLE MOTOR α*i*-B / β*i*-B series DESCRIPTIONS (B-65452EN)" or "FANUC AC SPINDLE MOTOR α*i* series PARAMETER MANUAL (B-65280EN)".

4044	Proportional gain of the velocity loop in servo mode (gear 1, gear 2)

4045	Proportional gain of the velocity loop in servo mode (gear 3, gear 4)

[Unit of data]
[Valid data range] 0 to 32767
Set a proportional gain for the velocity loop in a servo mode (such as rigid tapping mode).

4052	Integral gain of the velocity loop in the servo mode (gear 1, gear 2)

4053	Integral gain of the velocity loop in the servo mode(gear 3, gear 4)

[Unit of data]
[Valid data range] 0 to 32767
Set an integral gain of the velocity loop in a servo mode (such as rigid tapping mode).

4065	Position gain in the servo mode (HIGH)

4066	Position gain in the servo mode (MEDIUM HIGH)

4067	Position gain in the servo mode (MEDIUM LOW)

4068	Position gain in the servo mode (LOW)

[Unit of data] 0.01sec^{-1}
[Valid data range] 0 to 65535
Set a servo loop gain in a servo mode (such as rigid tapping mode).

⚠ **CAUTION**
Set a loop gain for spindle position control in rigid tapping using a serial spindle. In these parameters, basically, set the same values as those set in parameters Nos. 5281 to 5284 (loop gains for position control of the tapping axis).
Which serial spindle parameter (i.e., loop gain) is actually used to operate the spindle depends on the serial spindle clutch/gear selection signals CTH1 and CTH2. Accordingly, which parameter is to be used must be determined by considering the gear switching and PMC software.

The table below indicates the relationship between the spindle gear selection signals and selected gear numbers.

CTH1	CTH2	Gear selected	Parameter No. to be used		
0	0	HIGH	4065	4044	4052
0	1	MIDEUM HIGH	4066		
1	0	MIDEUM LOW	4067	4045	4053
1	1	LOW	4068		

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Rigid tapping
CONNECTION MANUAL (FUNCTION) (This manual)	Spindle speed control
	Multi spindle control
FANUC SERVO AMPLIFIER αi–B series DESCRIPTIONS (B-65412EN)	Rigid tapping
FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)	Rigid tapping
	Tuning way for FSSB High-speed Rigid Tapping

10.14 SPINDLE SYNCHRONOUS CONTROL

Overview

This function can exercise synchronous control on spindles. This function can also control the rotation phase of spindles, so that not only a round rod but also a non-standard workpiece can be grasped by either of two spindles.
A combination of a master spindle and slave spindle in spindle synchronous control can be selected arbitrarily from spindles.

This function can be used with serial spindles.

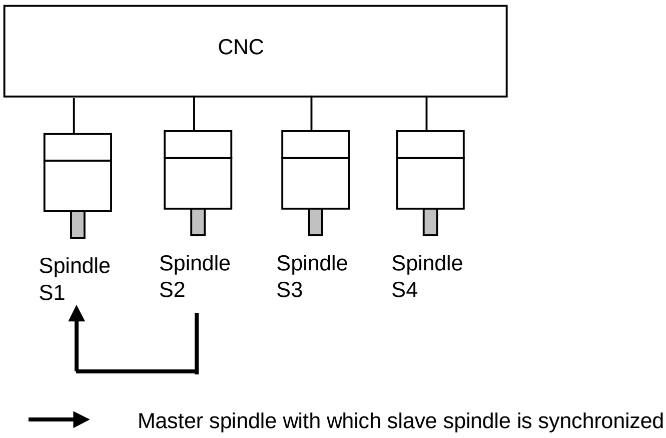
Explanation

- Synchronous-spindle configuration (when the one path system is used)

In spindle synchronous control, the spindle to which an S command is issued is called the master spindle. A spindle which ignores any S command that is issued for it, instead rotating synchronously with the master spindle, is called the slave spindle.

When the bit 4 (SSS) of parameter No.3704 is set to 0
The configuration of synchronized spindles is as follows:

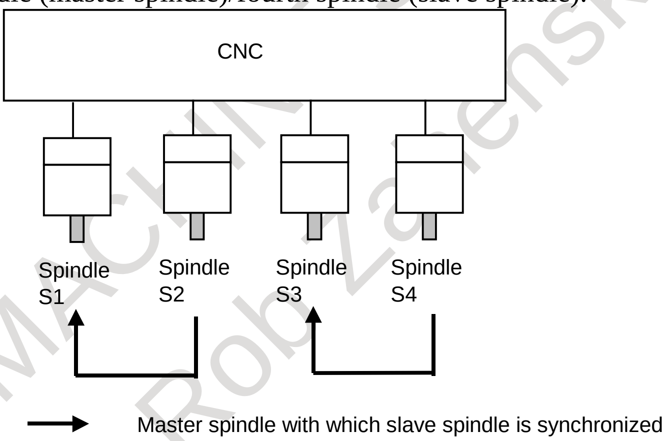
Master spindle	Slave spindle
First spindle	Second spindle



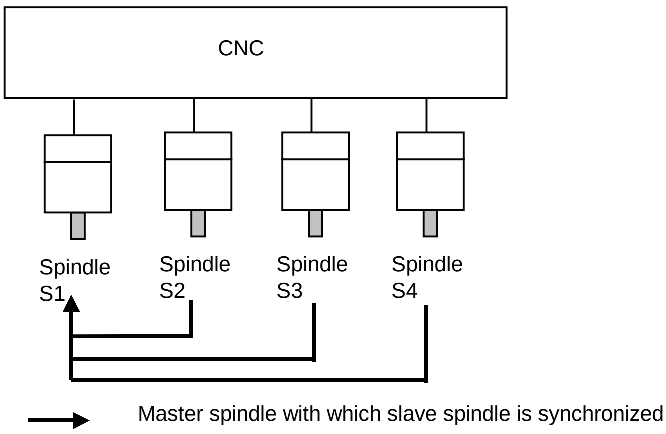
When the bit 4 (SSS) of parameter No. 3704 is se to 1
Which slave spindle is synchronized with which master spindle is set using parameter No. 4831.

Master spindle	Slave spindle
Arbitrary spindle	Arbitrary spindle

- When four spindles are involved, two combinations of spindles can be formed for spindle synchronization.
  Example)
  Two combinations are possible: first spindle (master spindle)/second spindle (slave spindle) and third spindle (master spindle)/fourth spindle (slave spindle).



- Multiple slave spindles can be synchronized with one master spindle.
  Example)
  A combination of first spindle (master spindle)/second, third, and fourth spindles (slave spindles) is possible.

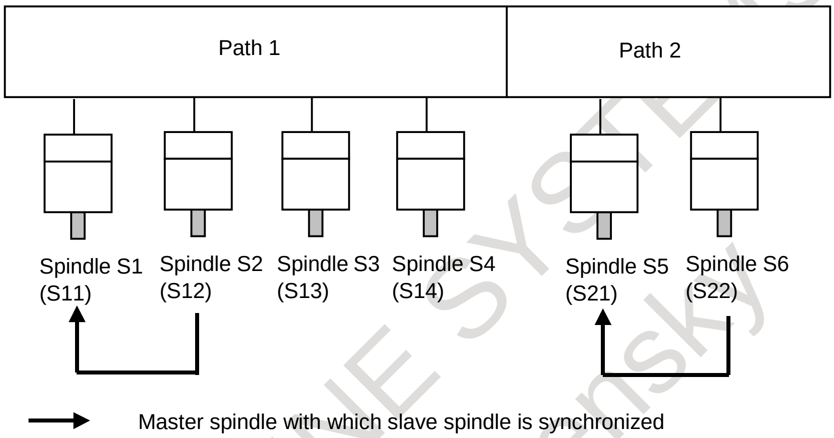


NOTE
1. A combination that uses a slave spindle as a master spindle is not allowed.
2. When multiple slave spindles are configured for synchronous control, a restriction is imposed on phase matching operation. See the description of the spindle phase synchronization control signal.

- Configuration of synchronized spindles (in a multi-path system)

- When the bit 4 (SSS) of parameter No.3704 is set to 0
  The configuration of synchronized spindles is as follows:

Master spindle	Slave spindle
First spindle of each path	Second spindle of each path



- When the bit 4 (SSS) of parameter No.3704 is set to 1
  Spindle synchronization can be performed by combining arbitrary spindles.
  Which slave spindle is synchronized with which master spindle is set using parameter No. 4831/4832.

Master spindle	Slave spindle
Arbitrary spindle of each path	Arbitrary spindle of each path

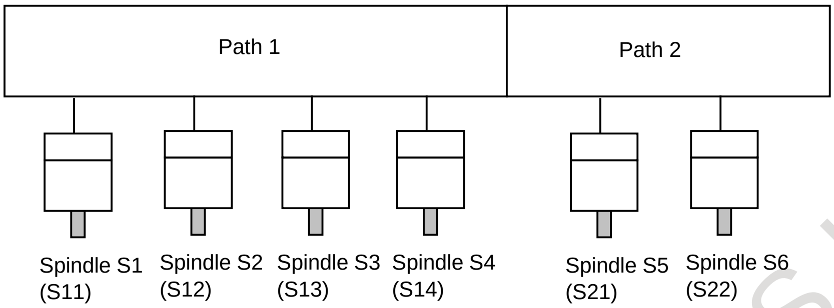
Set a master spindle for each slave in parameter No.4831.
As a master spindle, set a spindle number in the same path as for a slave spindle.

By setting a logic spindle number common to the system in parameter No.4832, such spindle synchronization that an arbitrary spindle belonging to a different path is used as a master spindle can be exercised.
The spindle number of a master spindle to be set is a logic spindle number common to the system. In this case, set 0 in parameter No.4831 for all.
For the path to which a slave spindle belongs and the path to which a master spindle belongs, bit 4 (SSS) of parameter No.3704 needs to be set to 1 (to enable spindle synchronization using arbitrary axes).

Example of parameter setting)
Spindle configuration of each path
Six spindles (four spindles of path 1 + two spindles of path 2)

Logic spindle number common to system	Path number + intra-path spindle number
First spindle (S1)	First spindle of path 1 (S11)
Second spindle (S2)	Second spindle of path 1 (S12)
Third spindle (S3)	Third spindle of path 1 (S13)
Fourth spindle (S4)	Fourth spindle of path 1 (S14)

Logic spindle number common to system	Path number + intra-path spindle number
Fifth spindle (S5)	First spindle of path 2 (S21)
Sixth spindle (S6)	Second spindle of path 2 (S22)

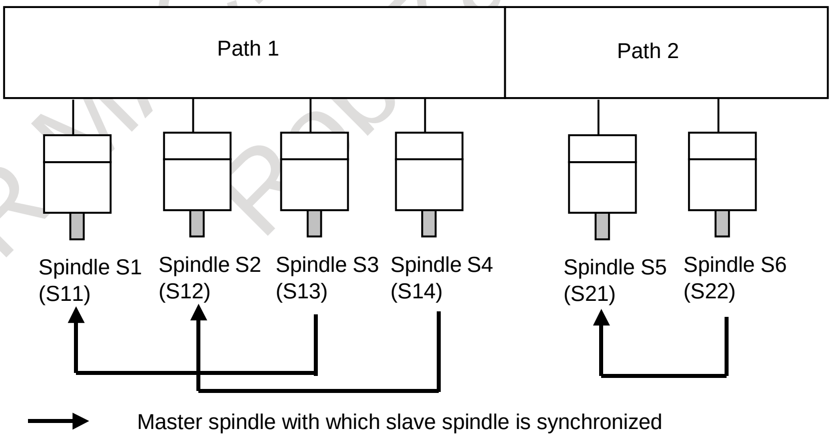


Example 1:

Combining spindles of the same path for spindle synchronization

Parameter setting
Bit 4 (SSS) of parameter No.3704 (PATH1)=1, (PATH2)=1
Parameter No.4831(S1) =0, (S2) =0,(S3) =1,(S4) =2,(S5) =0,(S6) =1
Parameter No.4832(ALL) =0
or
Parameter No.4831(ALL) =0
Parameter No.4832(S1) =0, (S2) =0,(S3) =1,(S4) =2,(S5) =0,(S6) =5

Master spindle	Slave spindle
First spindle of path 1 (S11)	Third spindle of path 1 (S13)
Second spindle of path 1 (S12)	Fourth spindle of path 1 (S14)
First spindle of path 2 (S21)	Second spindle of path 2 (S22)

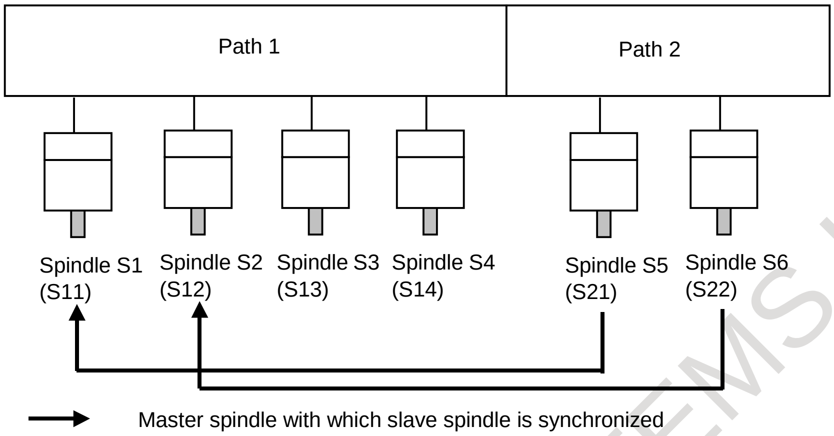


Example 2:

Combining spindles of different paths for spindle synchronization

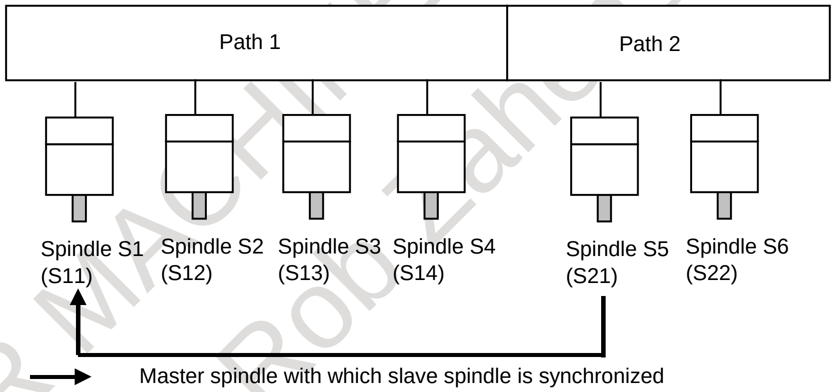
Parameter setting
Bit 4 (SSS) of parameter No.3704 (PATH1)=1, (PATH2)=1
Parameter No.4831(ALL) =0
Parameter No.4832(S1) =0, (S2) =0,(S3) =0,(S4) =0,(S5) =1,(S6) =2

Master spindle	Slave spindle
First spindle of path 1 (S11)	First spindle of path 2 (S21)
Second spindle of path 1 (S12)	Second spindle of path 2 (S22)



- When the bit 5 (SCB) of parameter No.4800 is set to 1
  The configuration of synchronized spindles is as follows (regardless of the setting of bit 4 (SSS) of parameter No.3704):

Master spindle	Slave spindle
First spindle of path 1	First spindle of path 2



As control signals, the 16TT system compatible signal interface becomes usable.

- Speed synchronization

- When the spindle synchronous control signal is set to “1”, the spindle synchronous control mode is set.
  When spindle synchronous control is specified, each spindle is accelerated or decelerated to a specified speed then enters the synchronous control state.
- The spindle synchronous speed control completion signal is output when the two spindles have reached the speed matching a specified spindle synchronous speed and the speed difference between the two spindles is within the value set in parameter No.4033.

- Phase matching

- Spindle phase synchronization is executed by setting the spindle phase synchronization control signal to 1 in the spindle synchronous control mode (after the output of the spindle synchronous speed control completion signal).
  The synchronous state between the two axes is not maintained during spindle phase synchronization operation (until the spindle phase synchronization control completion signal is set to 1).

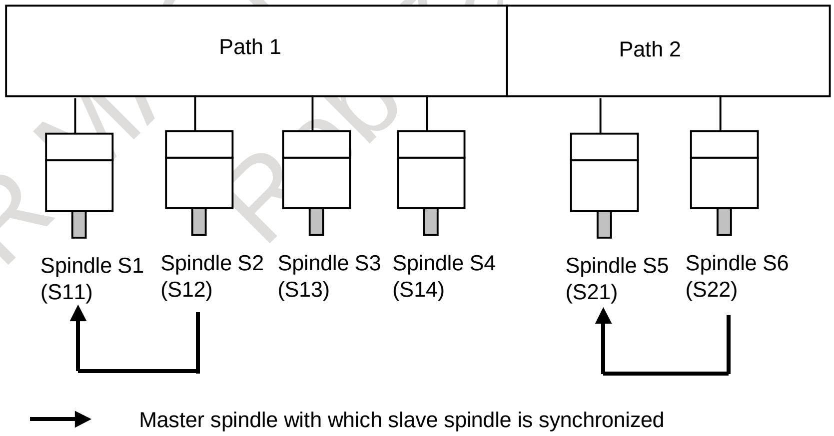
- The spindle phase synchronization completion signal is output when the error difference between the two spindles lies within the allowable number of pulses set in the NC parameter No. 4810.
  If spindle phase synchronization is specified after spindle phase synchronization completion, another spindle phase synchronization operation is performed.
  When the two spindles hold a workpiece, do not specify spindle phase synchronization.

- Speed specification

- In the spindle synchronous control mode, the specification of a speed for the master spindle is valid.
- In the spindle synchronous control mode, a speed for the master spindle can be specified in the same way as for an ordinary S command.
  Moreover, signals such as the PMC-based spindle output control signals (SIND, SSIN, SSGN, and R01I to R12I), *SSTP, and SOR are valid as usual.
  If a speed is specified using a PMC-based spindle output control command, however, the parameters Nos. 3741, 3742, 3743, and 3744 for specifying the maximum speeds of the individual spindle gears must be correctly set beforehand.
  When 0 is set in the parameter corresponding to a selected gear, the spindle does not make a synchronous rotation.
- An S command specified for a master spindle and a speed specified for PMC-based spindle output control before the spindle synchronous control mode is set are valid even in the spindle synchronous control mode.
  A speed command for a master spindle during spindle synchronous control is also valid as a command for the master spindle after spindle synchronous control is canceled.

- Speed ratio control

- Spindle synchronous control can be exercised to set a speed ratio of master spindle: slave spindle to 1:n.
  Speed ratio control is exercised in spindle synchronization by setting the spindle synchronous speed ratio control signal SBRT <Gn038.1> to “1” in the spindle synchronous control mode. At this time, the speed ratio of master spindle: slave spindle is 1:n (n: value set in the parameter No.7635).



Mater spindle : Speed ratio setting for the slave spindle 1:2
When the speed S100 is specified for master spindle S1
Spindle S1 : Spindle speed S100(1/min)
Spindle S2 : Placed in synchronous control state with spindle speed S200 (1/min).

If the speed of each slave spindle specified by multiplying the speed specified for the master spindle by a speed ratio exceeds the value set in the parameter No.7636, the speed of each slave spindle is clamped to a set maximum allowable speed. At the same time, the speed of the master spindle is clamped to the speed that maintains the speed ratio.
When the speeds of multiple slave spindles are clamped, all spindles are clamped to the speeds that can maintain the speed ratios for synchronization control.

When the speed of a slave spindle is clamped, the spindle synchronous speed ratio control clamp signal RSMAX <Fn065.2> is set to “1”.

- Supplement

- To monitor an error mutually between two spindles in the spindle synchronous control mode, the spindle phase error monitor signal SYCAL <Fn044.4> (SYCAL1 to SYCAL4 <Fn043.0> to <Fn043.3>) is available. A synchronization error between two spindles is monitored at all times. When a synchronization error exceeding the value (absolute number of error pulses) set in parameter No.4811 is detected, this signal is set to “1”. When the synchronization error does not exceed the value set in parameter No.4811, the signal is set to “0”.
- Even when two spindles hold a workpiece in the synchronous control state, constant surface speed control can be exercised. Even if the speed of a spindle changes, however, the synchronous speed changes within the parameter-set range (acceleration/deceleration time constant in spindle synchronous control).
- In spindle synchronous control, the spindle speed offset value (parameter No.3731) is invalid.

For details of synchronous-spindle connection, see the description of serial spindles.

Limitation

- When spindles are placed in a mode other than a spindle rotation mode such as the Cs contouring control mode, rigid tapping, and spindle command synchronous, the spindle synchronous control mode cannot be used. In the spindle rotation mode, switch the mode of the master and slave spindles to the spindle synchronous control mode.
- For spindles placed in the spindle synchronous control mode, commands for Cs contouring control, rigid tapping, spindle command synchronous, and so forth cannot be specified.
- The maximum speed in spindle synchronous control depends on the maximum speed (parameter No. 4020) of the spindle motor of the master spindle.

  Example)

  Maximum speed of the spindle motor of a master spindle: 6000 min^{-1}

  Maximum speed of the spindle motor of a slave spindle: 4500 min^{-1}

  In the example above, 6000 min^{-1} is specified as the maximum speed of the master spindle for 12bit = 4095. However, when 6000 min^{-1} is specified in spindle synchronous control, the excessive-speed alarm (spindle unit side alarm AL-07) is issued with the slave spindle.

  So, do not specify a speed exceeding 4500 min^{-1} in the example above.

Diagnosis data

0418	**Position deviation of each spindle**

When spindles are involved in a position loop, the position deviation of each spindle is indicated.
The unit of a detector used in each mode is used.

0425	**Synchronization error of each spindle**

The absolute value of the synchronization error of each spindle in spindle synchronization is indicated.
A synchronization error is indicated with a slave spindle.

Spindle tuning screen (spindle synchronous control)

In the spindle synchronous control mode, the spindle tuning screen displays the information below.
"Spindle position deviation" indicates the position deviation of the spindle currently displayed.
"Synchronous deviation " indicates the absolute value of a synchronization error in spindle synchronization. The value of a synchronization error is displayed on the spindle tuning screen for the slave spindle.
When bit 3 (SVP) of parameter No.3799 is set to 1, "synchronous deviation" indicates the peak hold value of a synchronization error.

For details, refer to Spindle tuning screen of MAINTENANCE MANUAL (B-64695EN).

Motor speed Spindle speed Spindle position deviation S1 Spindle position deviation S2 Synchronous deviation

Signal

Spindle synchronous control signal SPSYC<Gn038.2>

[Classification] Input signal

[Function] This signal specifies switching to the spindle synchronous control mode for the first/second spindles.

[Operation] When this signal is set to “1”, the spindle synchronous control mode for the first/second spindles is set.
When this signal is set to “0”, the spindle synchronous control mode for the first/second spindles is canceled.

This signal is valid only when bit 4 (SSS) of parameter No.3704 is set to 0.
When bit 5 (SCB) of parameter No.4800 is set to 1, an address of n = 0 is valid.

Spindle synchronous control signal of each spindle SPSYC1 to SPSYC4<Gn288.0 - Gn288.3>

[Classification] Input signal

[Function] These signals specify the switching of each spindle to the spindle synchronous control mode.

[Operation] When each of these signals is set to “1”, the spindle synchronous control mode with a spindle set as a slave spindle is set.
When each of these signals is set to “0”, the spindle synchronous control mode with a spindle set as a slave spindle is canceled.
When SPSYCs is set to “1”, spindle synchronization with a spindle set as a slave spindle is performed.

These signals are valid only when bit 4 (SSS) of parameter No.3704 is set to 1 and bit 5 (SCB) of parameter No.4800 is set to 0.
Which spindle is synchronized with which spindle is set using parameter No.4831 or 4832.

> **NOTE**
> For the correspondence between the signal of each spindle and actual signal address, see the description of "Signal address".

Spindle phase synchronization control signal SPPHS<Gn038.3>

[Classification] Input signal

[Function] This signal specifies the spindle phase synchronization control mode (phase matching) for the first/second spindles.

[Operation] When this signal makes a transition from “0” to “1”, spindle phase synchronization control for the first/second spindles is exercised.
This signal is valid when the spindle synchronous control signal SPSYC is set to “1”.
Specify this signal after the spindle synchronous speed control completion signal FSPSY is set to “1”.
A spindle phase synchronization control operation is started on the rising edge of this signal.

So, phase matching once performed is not lost when this signal is set to “0”.
However, when this signal is set to “1” from “0” again, a phase matching operation is performed.

This signal is valid only when bit 4 (SSS) of parameter No.3704 is set to 0.
Set a phase shift amount in parameter No.4034.
When bit 5 (SCB) of parameter No.4800 is set to 1, an address of n = 0 is valid.

> **NOTE**
> Before specifying this signal, ensure that the spindle synchronous speed control completion signal FSPSY is set to “1”. When the spindle synchronous speed control completion signal FSPSY is set to “0”, phase matching operation is not performed.

Spindle phase synchronization control signal of each spindle SPPHS1 to SPPHS4<Gn289.0 to Gn289.3>

[Classification] Input signal

[Function] These signals specify the spindle phase synchronization control mode (phase matching) for each spindle.

[Operation] When each of these signals is set to “1”, spindle phase matching is performed in the spindle synchronous control mode with a spindle set as a slave spindle.
These signals are valid when the spindle synchronous control signal SPSYCs for each spindle is set to “1”.
Specify each of these signals after the spindle synchronous speed control completion signal FSPSYs for each spindle is set to “1”.
A spindle phase synchronization control operation is started on the rising edge of each of these signals.
So, phase matching once performed is not lost when each of these signals is set to “0”.
However, when each of these signals is set to “1” from “0” again, a phase matching operation is performed.

These signals are valid only when bit 4 (SSS) of parameter No.3704 is set to 1 and bit 5 (SCB) of parameter No.4800 is set to 0.
Set a phase shift amount in parameter No.4034.

> **NOTE**
> 1 Before specifying these signals, ensure that the spindle synchronous speed control completion signal FSPSYx is set to “1”. When the spindle synchronous speed control completion signal FSPSYx is set to “0”, phase matching operation is not performed.
> 2 Phase matching in the state where spindles are rotating in synchronism is enabled only for one slave spindle synchronized with one master spindle. When phase matching is to be performed in the state where multiple slave spindles are rotating in synchronism with one master spindle, do not specify spindle phase synchronization control on multiple spindles.
> 3 Before performing phase matching of one master spindle with multiple slave spindles, ensure that all spindles are stopped.
> 4 For the correspondence between the signal of each spindle and actual signal address, see the description of "Signal address".

Spindle synchronous speed control completion signal FSPSY<Fn044.2>

[Classification] Output signal

[Function] This signal posts that spindle synchronous control (speed synchronization) for the first/second spindles is completed.

[Output cond.] This signal is set to “1” in the following case:

- This signal is output when two spindles have reached the speed equivalent to a specified spindle synchronous speed and the speed difference between the two spindles is equal to or less than the value set in parameter No.4033 in the spindle synchronous control mode.

This signal is set to “0” in the following cases:

- In the spindle synchronous control mode, two spindles have not reached the speed equivalent to a specified spindle synchronous speed.
- In the spindle synchronous control mode, the speed difference between two spindles is greater than the value set in parameter No.4033.
- The spindle synchronization control mode is not set.

This signal is valid only when bit 4 (SSS) of parameter No.3704 is set to 0.
When bit 5 (SCB) of parameter No.4800 is set to 1, an address of n = 0 is valid.

NOTE

Even if this signal is once set to “1”, this signal is set to “0” when the speed difference exceeds the value set in parameter No.4033 for a cause such as cutting load variation.

Spindle synchronous speed control completion signal for each spindle FSPSY1 to FSPSY4<Fn288.0 to Fn288.3>

[Classification] Output signal

[Function] These signals post that spindle synchronous control (speed synchronization) with each spindle set as a slave spindle is completed.

[Output cond.] This signal is set to “1” in the following case:

- This signal is output when two spindles have reached the speed equivalent to a specified spindle synchronous speed and the speed difference between the two spindles is equal to or less than the value set in parameter No.4033 in the spindle synchronous control mode.

This signal is set to “0” in the following cases:

- In the spindle synchronous control mode, two spindles have not reached the speed equivalent to a specified spindle synchronous speed.
- In the spindle synchronous control mode, the speed difference between two spindles is greater than the value set in parameter No.4033.
- The spindle synchronization control mode is not set.

These signals are valid only when bit 4 (SSS) of parameter No.3704 is set to 1 and bit 5 (SCB) of parameter No.4800 is set to 0.

NOTE

1 Even if this signal is once set to “1”, this signal is set to “0” when the speed difference exceeds the value set in parameter No.4033 for a cause such as cutting load variation.

2 For the correspondence between the signal of each spindle and actual signal address, see the description of "Signal address".

Spindle phase synchronization control completion signal FSPPH<Fn044.3>

[Classification] Output signal

[Function] This signal posts that spindle phase synchronization control (phase matching) for the first/second spindles is completed.

[Output cond.] This signal is set to “1” in the following case:

- This signal is output when phase matching is completed with the spindle phase synchronization control signal (when the error pulse difference between the two spindles is equal to or less than the value set in parameter No.4810) after the two spindles have reached the speed equivalent to a specified spindle synchronous speed in the spindle synchronous control mode.

This signal is set to “0” in the following cases:

- In the spindle synchronous control mode, the phase matching of two spindles is not completed.
- In the spindle synchronous control mode, the error pulse difference between two spindles is greater than the value set in parameter No.4810.
- The spindle synchronous control mode is not set.
- The spindle phase synchronization control mode is not set.

This signal is valid only when bit 4 (SSS) of parameter No.3704 is set to 0.
When bit 5 (SCB) of parameter No.4800 is set to 1, an address of n = 0 is valid.

> **NOTE**
> Even if this signal is once set to “1”, this signal is set to “0” when the speed difference exceeds the value set in parameter No.4810 for a cause such as cutting load variation.

Spindle phase synchronization control completion signal of each spindle FSPPH1 to FSPPH4<Fn289.0 to Fn289.3>

[Classification] Output signal

[Function] These signals post that spindle phase synchronization control (phase matching) with each spindle set as a slave spindle is completed.

[Output cond.] This signal is set to “1” in the following case:

- This signal is output when phase matching is completed with the spindle phase synchronization control signal (when the error pulse difference between the two spindles is equal to or less than the value set in parameter No.4810) after the two spindles have reached the speed equivalent to a specified spindle synchronous speed in the spindle synchronous control mode.

This signal is set to “0” in the following cases:

- In the spindle synchronous control mode, the phase matching of two spindles is not completed.
- In the spindle synchronous control mode, the error pulse difference between two spindles is greater than the value set in parameter No.4810.
- The spindle synchronous control mode is not set.
- The spindle phase synchronization control mode is not set.

These signals are valid only when bit 4 (SSS) of parameter No.3704 is set to 1 and bit 5 (SCB) of parameter No. 4800 is set to 0.

NOTE
1 Even if this signal is once set to “1”, this signal is set to “0” when the speed difference exceeds the value set in parameter No.4810 for a cause such as cutting load variation.
2 For the correspondence between the signal of each spindle and actual signal address, see the description of "Signal address".

Spindle phase error monitor signal SYCAL<Fn044.4>

[Classification] Output signal

[Function] This signal posts that in the spindle synchronous control mode for the first/second spindles, the error pulse difference between the two spindles is greater than a parameter-set value.

[Output cond.] This signal is set to “1” in the following case:

- In the spindle synchronous control mode, the error pulse difference between two spindles is greater than the value set in parameter No.4811 after spindle synchronous control is completed.

This signal is set to “0” in the following cases:

- The spindle synchronous control mode is not set.
- In the spindle synchronous control mode, the error pulse difference between two spindles is equal to or less than the value set in parameter No.4811.

This signal is valid only when bit 4 (SSS) of parameter No.3704 is set to 0.
When bit 5 (SCB) of parameter No.4800 is set to 1, an address of n = 0 is valid.

Spindle phase error monitor signal for each spindle SYCAL1 to SYCAL4<Fn043.0 to Fn043.3>

[Classification] Output signal

[Function] These signals post that in the spindle synchronous control mode with each spindle set as a slave spindle, the error pulse difference between two spindles is greater than a parameter-set value.

[Output cond.] This signal is set to “1” in the following case:

- In the spindle synchronous control mode, the error pulse difference between two spindles is greater than the value set in parameter No.4811 after spindle synchronous control is completed.

This signal is set to “0” in the following cases:

- The spindle synchronous control mode is not set.
- In the spindle synchronous control mode, the error pulse difference between two spindles is equal to or less than the value set in parameter No.4811.

These signals are valid only when bit 4 (SSS) of parameter No.3704 is set to 1 and bit 5 (SCB) of parameter No.4800 is set to 0.

NOTE
For the correspondence between the signal of each spindle and actual signal address, see the description of "Signal address".

Spindle synchronous speed ratio control signal SBRT<Gn038.1>

[Classification] Input signal

[Function] This signal specifies switching to speed ratio control in spindle synchronous control.

[Operation] When this signal is set to “1”, speed ratio control is set.
When this signal is set to “0”, speed ratio control is canceled.

This signal is common to all slave spindles that belong to the same path.

When bit 5 (SCB) of parameter No.4800 is set to 1, an address of n = 0 is valid.

Spindle synchronous speed ratio control clamp signal RSMAX<Fn065.2>

[Classification] Output signal

[Function] This signal posts the rotation state of a slave spindle under speed ratio control in spindle synchronous control.

[Output cond.] This signal is set to “1” in the following case:

- The speed of a slave spindle is clamped to the value set in the parameter No.7636 under speed ratio control in spindle synchronous control.

This signal is set to “0” in the following cases:

- The speed of any slave spindle is not clamped to the value set in the parameter No. 7636 under speed ratio control in spindle synchronous control.
- The spindle synchronization control mode is not set.

This signal is common to all slave spindles that belong to the same path.
When bit 5 (SCB) of parameter No.4800 is set to 1, an address of n = 0 is valid.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn038					SPPHS	SPSYC	SBRT	
Gn288					SPSYC4	SPSYC3	SPSYC2	SPSYC1
Gn289					SPPHS4	SPPHS3	SPPHS2	SPPHS1
Fn044				SYCAL	FSPPH	FSPSY		
Fn065						RSMAX		
Fn288					FSPSY4	FSPSY3	FSPSY2	FSPSY1
Fn289					FSPPH4	FSPPH3	FSPPH2	FSPPH1
Fn043					SYCAL4	SYCAL3	SYCAL2	SYCAL1

- Signal allocation of spindle-type signals

The signal allocation of spindle-type varies according to the path number to which each spindle belongs and intra-path spindle number.

Here, the address allocation of the spindle synchronous control signal for each spindle SPSYCs, which is a spindle-type signal, is described below.

- When the one path system is used :
  When four spindles of path 1 are used

Logic spindle number	Path number + intra-path spindle number	Signal symbol	Signal address
First spindle (S1)	First spindle of path 1 (S11)	SPSYC1	G0288.0
Second spindle (S2)	Second spindle of path 1 (S12)	SPSYC2	G0288.1
Third spindle (S3)	Third spindle of path 1 (S13)	SPSYC3	G0288.2
Fourth spindle (S4)	Fourth spindle of path 1 (S14)	SPSYC4	G0288.3

- When the multi path system is used :
  When six spindles consisting of three spindles of path 1 and three spindles of path 2 are used

Logic spindle number	Path number + intra-path spindle number	Signal symbol	Signal address
First spindle (S1)	First spindle of path 1 (S11)	SPSYC1	G0288.0
Second spindle (S2)	Second spindle of path 1 (S12)	SPSYC2	G0288.1
Third spindle (S3)	Third spindle of path 1 (S13)	SPSYC3	G0288.2
Fourth spindle (S4)	First spindle of path 2 (S21)	SPSYC1	G1288.0
Fifth spindle (S5)	Second spindle of path 2 (S22)	SPSYC2	G1288.1
Sixth spindle (S6)	Third spindle of path 2 (S23)	SPSYC3	G1288.2

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3799					SVPs			

[Input type] Parameter input
[Data type] Bit spindle

#3 SVPs As synchronization errors displayed on the spindle tuning screen:
0: Monitor values are displayed.
1: Peak-hold values are displayed.
Spindle synchronization errors are displayed on the side of the spindle that functions as a slave axis in spindle synchronization control.

	#7	#6	#5	#4	#3	#2	#1	#0
3704				SSS				

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#4 SSS Synchronous spindle control by each spindle is:
0: Not performed.
1: Performed.
The master axis and slave axis of synchronous spindle control can be selected from the arbitrary spindles.
The target spindle of synchronous spindle control is specified in parameter No.4831.
In addition, the following signals affect the control.
- Synchronous spindle signals of each spindle SPSYCs
- Signals of synchronous control of the spindle phase for each spindle SPPHSs

	#7	#6	#5	#4	#3	#2	#1	#0
3716								A/Ss

[Input type] Parameter input
[Data type] Bit spindle

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 A/Ss Spindle motor type is :
0: Analog spindle.
1: Serial spindle.

3717	Spindle amplifier number to each spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled axes
Set a spindle amplifier number to be assigned to each spindle.
0: No spindle amplifier is connected.
1: Spindle motor connected to amplifier number 1 is used.
2: Spindle motor connected to amplifier number 2 is used.
to
n: Spindle motor connected to amplifier number n is used.

4032	Acceleration / deceleration time constant for spindle synchronous control

[Input type] Parameter input
[Data type] Word spindle
[Data unit] $1min^{-1}/sec$
[Valid data range] 0 to 32767
This parameter sets an acceleration value for linear acceleration/deceleration when the synchronous speed command for spindle synchronous control is changed.

> **NOTE**
> 1 Set exactly the same data for a master axis spindle and a slave axis. When different data is set, synchronization between the two spindles is not guaranteed.
> 2 When this parameter is set to 0, acceleration/deceleration of a spindle is not applied. Set an appropriate value.

4033	Detection level for spindle synchronous speed

[Input type] Parameter input
[Data type] Word spindle
[Data unit] $1min^{-1}/sec$
[Valid data range] 0 to 32767
For the synchronous speed command at spindle synchronous control, if the error of the respective spindle motor speeds are within the setting level, the spindle synchronous control complete signal (FSPSY) becomes “1”.

4034	The shift amount in spindle phase synchronization control

[Input type] Parameter input
[Data type] Word spindle
[Data unit] 1 pulse unit (360°/4096)
[Valid data range] 0 to 4095

Sets the shift amount from the reference point at spindle phase synchronous control (one-rotation signal).

	#7	#6	#5	#4	#3	#2	#1	#0
4800			SCB	SYM				

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#4 SYM In spindle synchronous control, spindle maximum speed uses:
0: the one of a master axis.
1: low one either a master axis or a slave axis.

NOTE
This parameter is valid only for T type gear selection method.

#5 SCB The combination of a master spindle and slave spindle for spindle synchronization control depends on:
0: Setting of bit 4 (SSS) of parameter No.3704.
When bit 4 (SSS) of parameter No.3704 is set to 0
The first spindle and second spindle of each path can be selected as the master spindle and slave spindle, respectively, for spindle synchronization.
When bit 4 (SSS) of parameter No.3704 is set to 1
A combination of arbitrary spindles of each path can be selected for spindle synchronization.
Set a master spindle for each slave spindle in parameter No.4831. Set a spindle number of each path.
By setting a spindle number common to the system in parameter No.4832, an arbitrary spindle that belongs to a different path can be selected as a master spindle for spindle synchronization. Set a spindle number common to the system. Set parameter No.4831 to 0. Spindle synchronization based on arbitrary spindles must be enabled for the path to which a slave spindle belongs and for the path to which a master spindle belongs.
1: Conventional 16TT system compatible specifications.
The first spindle of path 1 and the first spindle of path 2 can be selected as the master spindle and slave spindle, respectively, for spindle synchronization.
As control signals, the signal interface of the 16TT system compatible specifications can be used.

	#7	#6	#5	#4	#3	#2	#1	#0
4801								SNDs

[Input type] Parameter input
[Data type] Bit spindle

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 SNDs During spindle synchronization control, the rotation direction of each spindle motor is:
0: Same as the specified sign.
1: Opposite to the specified sign.

	#7	#6	#5	#4	#3	#2	#1	#0
4809								**NSY**

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 NSY When the spindle speed changes during spindle synchronization control, resolution improvement is:
0: Disabled. (Speed change in units of about 3.7 [min^{-1}])
1: Enabled. (Speed change in units of about 0.03 [min^{-1}] at minimum but not higher than maximum spindle speed/4095 [min^{-1}])

This parameter is valid when spindle synchronization control or spindle-spindle polygon turning is used.
Using high-precision spindle speed control and spindle synchronization control simultaneously requires setting the parameter to 1.

> **NOTE**
> Using this function requires the serial spindle software that supports it.

4810	**Error pulse between two spindles when synchronizing phases in the spindle synchronization control mode**

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 255

Set an allowable error pulse value between two spindles at phase synchronization time in the spindle synchronization control mode.
This parameter is used to check the completion of phase synchronization performed in the spindle synchronization control mode and to check the phase difference during spindle synchronization control.
When the error pulse value between two spindles become equal to or less than the value set in this parameter, the spindle phase synchronization control completion signals FSPPH<F044.3> and FSPPH1 to 4<F289.0 to 3> are set to “1”.

4811	**Allowable error count for the error pulses between two spindles in the spindle synchronization control mode**

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 32767

Set the allowable error count for the error pulses between two spindles in the spindle synchronization control mode.

This parameter is used to check a spindle synchronization error phase difference.
When a spindle synchronization error equal to or greater than the value set in this parameter is detected, the phase error monitor signals SYCAL<Fn044.4> and SYCAL1 to 4<Fn043.0 to Fn043.3> are set to “1”.

4831	Master axis of each slave spindle under spindle synchronous control

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled spindle axes (within a path)

When a spindle is set as a slave spindle in spindle synchronization control on each spindle, set which spindle (master spindle) the slave spindle is to be synchronized with.
[Examples of parameter setting]

- When spindle synchronization control is exercised with the first spindle selected as a master spindle and the second spindle selected as a slave spindle:
  No. 4831(1)=0
  No. 4831(2)=1
  No. 4831(3)=0
  No. 4831(4)=0

- When spindle synchronization control is exercised with four spindles under the following combinations:
  (Two combinations, namely, first spindle (master spindle)/second spindle (slave spindle), and third spindle (master spindle)/fourth spindle (slave spindle))
  No. 4831(1)=0
  No. 4831(2)=1
  No. 4831(3)=0
  No. 4831(4)=3

- When spindle synchronization control is exercised with one master spindle and multiple slave spindles:
  (First spindle (master spindle)/second spindle (slave spindle)/third spindle (slave spindle)/fourth spindle (slave spindle))
  No. 4831(1)=0
  No. 4831(2)=1
  No. 4831(3)=1
  No. 4831(4)=1

> **NOTE**
> 1 This parameter is valid only if bit 4 (SSS) of parameter No.3704 is set to 1.
> 2 The setting of a slave spindle as a master spindle is invalid.
> This parameter must be set to 0 for a spindle to be set as a master spindle.
> 3 In this parameter, set a spindle number within the same path.
> When a spindle not belonging to the local path is to be selected as a master spindle for spindle synchronization, set a spindle number common to the system in parameter No.4832. In such a case, set 0 in this parameter.

4832	Master spindle of each slave spindle under spindle synchronization control (spindle number common to the system)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled spindle axes (common to the system)

When a spindle is set as a slave spindle in spindle synchronization control on each spindle, set which spindle (master spindle) the slave spindle is to be synchronized with.

NOTE
1. This parameter is valid if bit 4 (SSS) of parameter No.3704 is set to 1.
   Bit 4 (SSS) of parameter No.3704 must be set to 1 (to enable spindle synchronization based on arbitrary spindles) for the path to which a slave spindle belongs and for the path to which a master spindle belongs.
2. The setting of a slave spindle as a master spindle is invalid.
   This parameter must be set to 0 for a spindle to be set as a master spindle.
3. In this parameter, set a spindle number common to the system.
4. When this parameter is used, parameter No.4831 is set to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
7603						SBR		

[Input type] Parameter input
[Data type] Bit path

#2 SBR For spindle synchronization control, speed ratio control is:
0: Not used.
1: Used.

7635	Ratio of slave spindle speed in spindle synchronization control

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 9

This parameter sets the ratio of master spindle speed: slave spindle speed (1:n) in spindle synchronization control.

NOTE
This parameter is valid only when bit 2 (SBR) of parameter No. 7603 is set to 1.

7636	Maximum allowable slave spindle speed in spindle synchronization control

[Input type] Parameter input
[Data type] Word spindle

[Unit of data] min^{-1}
[Valid data range] 0 to 19999

The speed of the slave spindle under speed ratio control in spindle synchronization control is clamped so that the speed does not exceed the value set in this parameter.

> **NOTE**
> 1 This parameter is valid only when bit 2 (SBR) of parameter No. 7603 is set to 1.
> 2 When speed ratio control in spindle synchronization control is used, be sure to set this parameter.
> 3 When 0 is set, the speed is clamped to 0, disabling rotation under spindle synchronization.

Alarm and message

Number	Message	Description
PS0194	SPINDLE COMMAND IN SYNCHRO-MODE	A Cs contour control mode, spindle positioning command, or rigid tapping mode was specified during the spindle synchronous control mode or spindle command synchronous control mode.

Caution

> ⚠ **CAUTION**
> 1 Phase error monitor signal SYCAL is used for monitoring a phase shift in spindle synchronous control. The processing performed when a phase shift is detected depends on the specifications determined by the machine tool builder.
> 2 A gear ratio of 1:1 only is allowed between the spindle and position coder.
> 3 A CNC system restriction is imposed on the maximum number of spindles.

Reference item

For details of the parameters Nos.4000 to 4539, signals, and alarms for the serial spindle control unit, refer to the manuals related to serial spindles listed below.
FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)

10.14.1 Arbitrary Spindle Position Phase Synchronization Function

Overview

In spindle synchronous control, when workpiece whose cross section is non-circular is gripped between master spindle and slave spindle, phase of master and slave has to be synchronized. In order to synchronize the phase of master spindle and slave spindle by spindle phase synchronous control so far, phase difference between master spindle and slave spindle has to measure by hand, and set to parameter.
This function can detect master and slave spindle position automatically by setting DI signal, and save spindle positions to parameters. In spindle phase synchronous control, phase can synchronize by loading saved spindle positions. Therefore, setup time for spindle phase synchronous control can be shortened.
Besides, master and slave spindle position can save up to 4 pairs. DI signal can select which of spindle position pairs will be loaded. Therefore, plural non-circular workpieces can be applied.
This function can use with serial spindles.

Explanation

Save of spindle position

In order to grip workpiece whose cross section is non-circular between master spindle and slave spindle, master and slave spindle positions are detected automatically, and saved to parameters Nos.4840 to 4843.

Master and slave spindle position can save up to 4 pairs. Master and slave spindle position are saved to parameters Nos.4840 to 4843 by the following procedure.

(1) Detect one-rotation signal of both master spindle and slave spindle beforehand. Check whether one-rotation signal has been detected or not by position coder one-rotation signal detection status signal PC1DT <Fn047.0, Fn051.0, Fn170.0, Fn268.0>.
(2) Command “S0 ;” to both master spindle and slave spindle so as not to rotate spindles.
(3) Turn off excitation of master spindle and slave spindle.
(4) Set a workpiece whose cross section is non-circular with master spindle and slave spindle.
(5) Turn on excitation of master spindle and slave spindle.
(6) Set spindle position save start signal SPMST1 to SPMST4 <Gn587.0 to Gn587.3> of slave spindle to “1”. Spindle position can save whether the spindle is stopping or rotating. If spindle position is saved during rotating spindles by spindle synchronous control, set spindle position save start signal SPMST1 to SPMST4 <Gn587.0 to Gn587.3> to “1” after having become spindle synchronous speed control completion signal FSPSY <Fn044.2> (or <Fn288.0 to Fn288.3>) “1”.
(7) When spindle position save start signal SPMST1 to SPMST4 <Gn587.0 to Gn587.3> is set from “0” to “1”, position coder one-rotation signal detection status signals PC1DT <Fn047.0, Fn051.0, Fn170.0, Fn268.0> of both master spindle and slave spindle are checked. If position coder one-rotation signal detection status signal <Fn047.0, Fn051.0, Fn170.0, Fn268.0> of master spindle or slave spindle is “0”, spindle position save error signal SPMER1 to SPMER4 <Fn577.4 to Fn577.7> becomes “1” because spindle position cannot be detected.
(8) When spindle position save start signal SPMST1 to SPMST4 <Gn587.0 to Gn587.3> is set from “0” to “1”, spindle position saving parameter Nos.4840 to No.4843 is selected by setting spindle position save selection signal SMSL11 to SMSL14 <Gn588.0 to Gn588.3>, SMSL21 to SMSL24 <Gn588.4 to Gn588.7> of slave spindle.
(9) Spindle positions of both master spindle and slave spindle are saved to parameter selected by (8), and spindle position save completion signal SPMFN1 to SPMFN4 <Fn577.0 to Fn577.3> of slave spindle becomes “1”. Besides, one of bit 0 to 3 (SM1 to SM4) of parameter No.4803 becomes 1.

Table 10.14.1 (a) Relationship between spindle position save selection signal and spindle position saving parameter

spindle position saving parameter	parameter notifying spindle position save is completed or not	Spindle position save selection signal	
		SMSL21 toSMSL24 <Gn588.4 toGn588.7>	SMSL11 toSMSL14 <Gn588.0 to Gn588.3>
No.4840	No.4803#0	0	0
No.4841	No.4803#1	0	1
No.4842	No.4803#2	1	0
No.4843	No.4803#3	1	1

(10) Set spindle position save start signal SPMST1 to SPMST4 <Gn587.0 to Gn587.3> to “0” after having become spindle position save completion signal SPMFN1 to SPMFN4 <Fn577.0 to Fn577.3> “0”.
(11) If spindle position save start signal SPMST1 to SPMFN4 <Fn577.0 to Fn577.3> is set to “0”, spindle position save completion signal SPMFN1 to SPMFN4 <Fn577.0 to Fn577.3> becomes “0”.

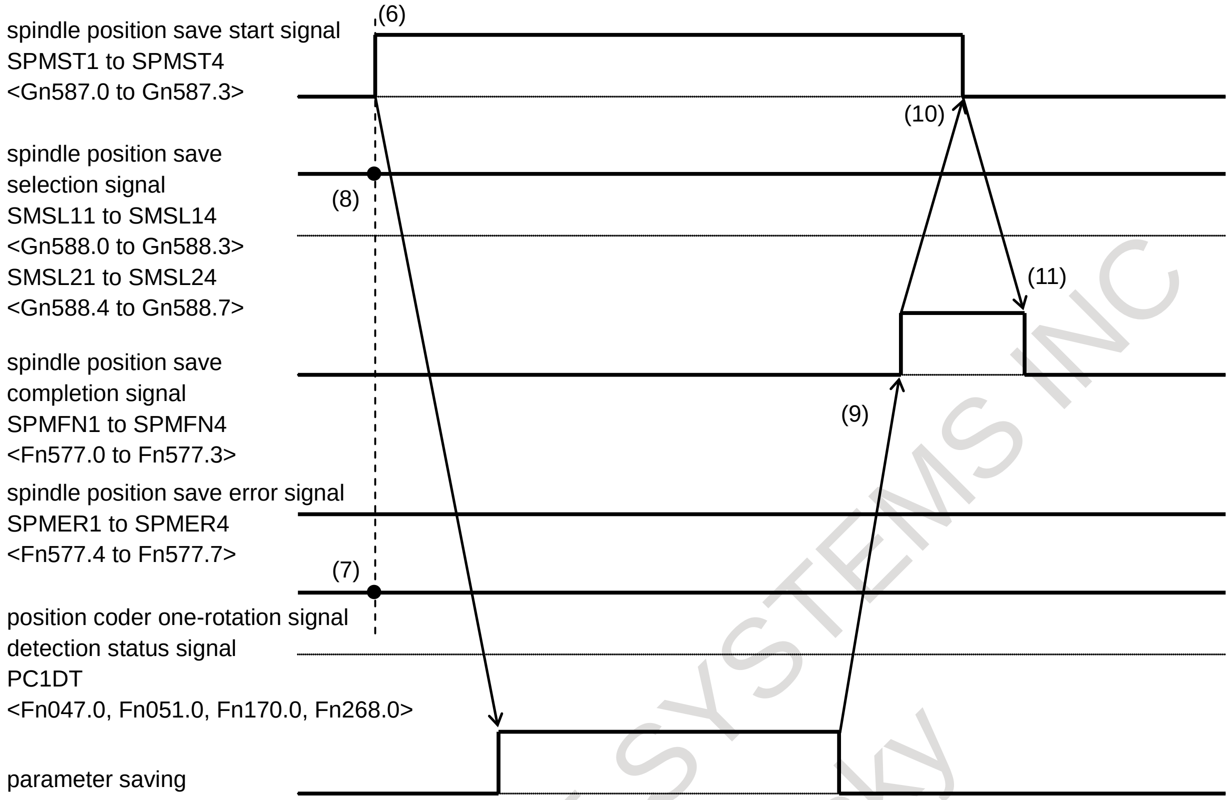


Fig.10.14.1 (a) Spindle position save

NOTE

Set bit 0 to 3 (SM1 to SM4) of parameter No.4803 to 0 if spindle position has to be saved again, for example, motor or detector is exchanged, parameter file of other machine is inputted, and so on.

Spindle position save error signal SPMER1 to SPMER4 <Fn577.4 to Fn577.7> of slave spindle becomes “1” in the following cases, because spindle position cannot be saved.

- Position coder one-rotation signal detection status signal PC1DT<Fn047.0, Fn051.0, Fn170.0, Fn268.0> of master spindle or slave spindle is ”0”.
- Spindle position save start signal SPMST1 to SPMST4 <Gn587.0 to Gn587.3> is set to “0” before having become spindle position save completion signal SPMFN1 to SPMFN4 <Fn577.0 to Fn577.3> “1”.
- Spindle position save to a parameter cannot be completed correctly.
- Spindle positions save start signal SPMST1 to SPMST4 <Gn587.0 to Gn587.3> of a slave spindle is set to “1” during saving spindle position of other master-slave pair (in the case of plural slave spindles for master spindle).

If spindle position save error signal SPMER1 to SPMER4 <Fn577.4 to Fn577.7> becomes “1”, bit 0 to 3 (SM1 to SM4) of parameter No.4803 corresponding to bit position of the signal becomes 1. If spindle position save error signal SPMER1 to SPMER4 <Fn577.4 to Fn577.7> becomes “1”, spindle position save start signal SPMST1 to SPMST4 <Gn587.0 to Gn587.3> has to be set to “0”. If spindle position save start signal SPMST1 to SPMST4 <Gn587.0 to Gn587.3> is set to “1” again, spindle position save error signal SPMER1 to SPMER4 <Fn577.4 to Fn577.7> becomes “0”.

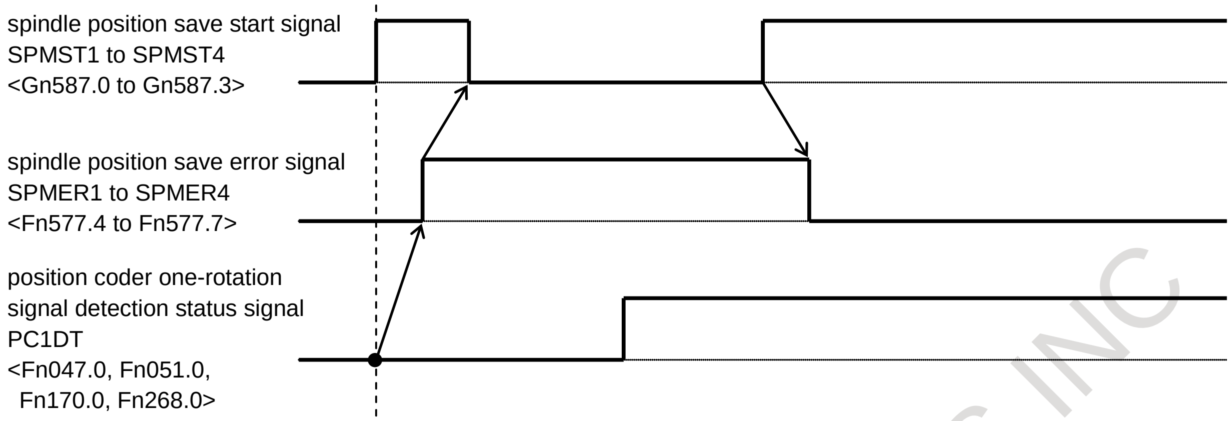


Fig.10.14.1 (b) Example of becoming spindle position save error signal “1”

Arbitrary spindle position phase synchronous control

In spindle phase synchronous control, phase between master spindle and slave spindle is synchronized based on parameter No.4034 (shift amount for spindle phase synchronous control). This function can use spindle position saved to parameter Nos.4840 to 4843 as phase shift amount (The value of parameter (No.4034) (shift amount for spindle phase synchronous control) is disabled for this function.). Up to 4 spindle positions can be memorized. Phase adjustment is performed for various non-standard workpieces. If spindle phase synchronous control signal SPPHS <Gn038.3> (or <Gn289.0 to Gn289.3>) is set to “1” with arbitrary spindle position phase synchronization control signal SPAPH1 to SPAPH4 <Gn587.4 to Gn587.7> of slave spindle “1”, phase synchronization whose shift amount is parameter Nos.4840 to 4843 is started. Spindle position save selection signal SMSL11 to SMSL14 <Gn588.0 to Gn588.3>, SMSL21 to SMSL24 <Gn588.4 to Gn588.7> of slave spindle can select which of parameter Nos.4840 to 4843 will be used. Relationship between spindle position save selection signal SMSL11 to SMSL14 <Gn588.0 to Gn588.3>, SMSL21 to SMSL24 <Gn588.4 to Gn588.7> and parameter Nos.4840 to 4843 is shown at Table 10.14.1 (a). If spindle position is not saved, and selected the pair of spindle position, alarm SP1256, “SPINDLE PHASE-SYNC IMPOSSIBLE” is issued to master spindle and slave spindle when arbitrary spindle position phase synchronous control is started. Bit 0 to 3 (SM1 to SM4) of parameter No.4803 of slave spindle can check whether spindle position save has been completed or not.

NOTE

1 Set bit 0 to 3 (SM1 to SM4) of parameter No.4803 to 0 if spindle position has to be saved again, for example, motor or detector is exchanged, parameter file of other machine is inputted, and so on.
2 Set bit 0 to 3 (SM1 to SM4) of parameter No.4803 to 0 if the saved spindle position (parameter Nos. 4840 to 4843) will not be used.
3 If bit 0 to 3 (SM1 to SM4) of parameter No.4803 is set to 1 by MDI unit or G10 command, alarm SP1256, “SPINDLE PHASE-SYNC IMPOSSIBLE” is not issued. Don’t set bit 0 to 3 (SM1 to SM4) of parameter No.4803 to 1 by MDI unit or G10 command, if spindle position of phase synchronization is not saved to parameter Nos.4840 to 4843.

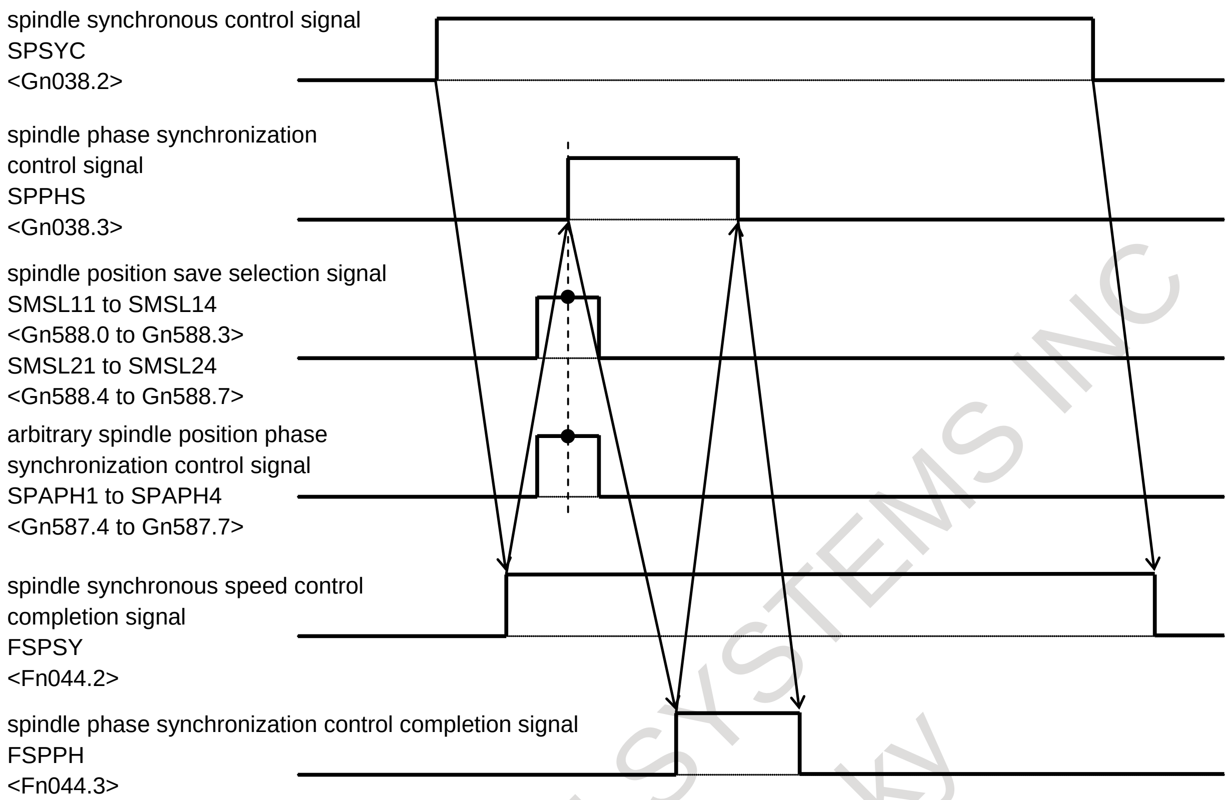


Fig.10.14.1 (c)　Arbitrary spindle position phase synchronous control

Limitation

Plural slave spindles

If there are plural slave spindles for master spindle, 3 or more spindles are synchronized as shown in Fig.10.14.1 (d), this function cannot be used.

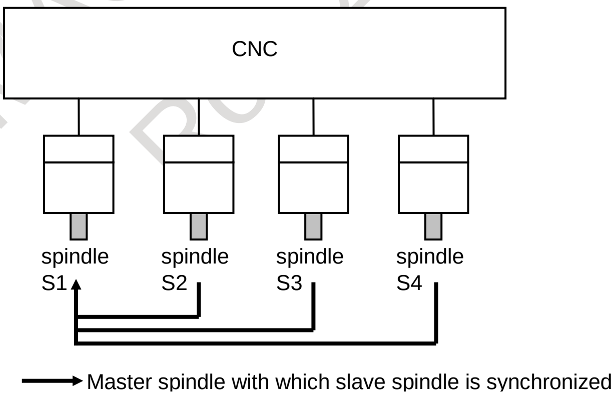


Fig.10.14.1 (d)　Example of plural slave spindles

Spindle synchronous control for spindle control with servo motor

This function cannot be used for spindle synchronous control for spindle control with servo motor.

Signal

Spindle position save start signal SPMST1 to SPMST4 <Gn587.0 to Gn587.3>

[Classification] Input signal

[Function] Spindle position save is started.

[Operation] If this signal of slave spindle is set to “1”, spindle position save of both master spindle and slave spindle is started.

Spindle position save selection signal SMSL11 to SMSL14 <Gn588.0 to Gn588.3> SMSL21 to SMSL24 <Gn588.4 to Gn588.7>

[Classification] Input signal

[Function] Parameter which saves spindle position is selected. Besides, parameter which is used as phase shift amount for spindle phase synchronous control is selected.

[Operation] When spindle position save start signal SPMST is from "0" to "1", parameter which saves spindle position is selected by setting this signal of slave spindle as shown in Table 10.14 (b).

Table 10.14.1 (b) Relationship between spindle position save selection signal and spindle position saving parameter

spindle position saving parameter	parameter notifying spindle position save is completed or not	Spindle position save selection signal	
		SMSL21 to SMSL24 <Gn588.4 to Gn588.7>	SMSL11 to SMSL14 <Gn588.0 to Gn588.3>
No.4840	No.4803#0	0	0
No.4841	No.4803#1	0	1
No.4842	No.4803#2	1	0
No.4843	No.4803#3	1	1

Besides, parameter which is used for spindle phase synchronous control with spindle position memorized in the parameter as phase shift amount is selected as shown in Table 10.14 (b).

Arbitrary spindle position phase synchronization control signal SPAPH1 to SPAPH4 <Gn587.4 to Gn587.7>

[Classification] Input signal

[Function] Spindle phase synchronous control whose phase shift amount is spindle position saved to parameter is executed.

[Operation] If spindle phase synchronous control signal SPPHS is set to "1" with this signal of slave spindle "1", spindle phase synchronous control whose phase shift amount is spindle position saved to parameter is started.

Spindle position save completion signal SPMFN1 to SPMFN4 <Fn577.0 to Fn577.3>

[Classification] Output signal

[Function] These signals notify whether spindle position save is completed or not.

[Output cond.] These signals become "1" under the following condition:

- Spindle position is saved to parameter.

These signals become "0" under the following condition:

- Spindle position save is completed, and spindle position save start signal SPMST is set to "0".
- Spindle position save is not executed.

Spindle position save error signal SPMER1 to SPMER4 <Fn577.4 to Fn577.7>

[Classification] Output signal

[Function] These signals notify the error of spindle position save to have occurred.

[Output cond.] These signals become "1" under the following condition:

- Spindle position save is started when position coder one-rotation signal detection status signal PC1DT of master spindle or slave spindle is "0".
- Spindle position save start signal SPMST is set to "0" before having become spindle position save completion signal SPMFN "1"
- Spindle position save cannot be completed correctly.

- Spindle positions save start signal SPMST1 to SPMST4 of a slave spindle is set to “1” during saving spindle position of other master-slave pair (in the case of plural slave spindles for master spindle).

These signals become “0” under the following condition:

- Spindle position save start signal SPMST is set to “0” after having become spindle position save error signal SPMER “1”, and spindle position save start signal SPMST is set to “1” again.
- Spindle position save is not executed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn587	SPAPH4	SPAPH3	SPAPH2	SPAPH1	SPMST4	SPMST3	SPMST2	SPMST1

	#7	#6	#5	#4	#3	#2	#1	#0
Gn588	SMSL24	SMSL23	SMSL22	SMSL21	SMSL14	SMSL13	SMSL12	SMSL11

	#7	#6	#5	#4	#3	#2	#1	#0
Fn577	SPMER4	SPMER3	SPMER2	SPMER1	SPMFN4	SPMFN3	SPMFN2	SPMFN1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
4803					SM4	SM3	SM2	SM1

[Input type] Parameter input
[Data type] Bit spindle

#0 SM1 Spindle position save to parameter No.4840 is:
0: Not completed.
1: Completed.

#1 SM2 Spindle position save to parameter No.4841 is:
0: Not completed.
1: Completed.

#2 SM3 Spindle position save to parameter No.4842 is:
0: Not completed.
1: Completed.

#3 SM4 Spindle position save to parameter No.4843 is:
0: Not completed.
1: Completed.

NOTE
Set bit 0 to 3 (SM1 to SM4) of parameter No.4803 to 0 if spindle position has to be saved again, for example, motor or detector is exchanged, parameter file of other machine is inputted, and so on.

4840	Spindle position 1 used as phase shift amount

4841	Spindle position 2 used as phase shift amount

4842	Spindle position 3 used as phase shift amount

4843	Spindle position 4 used as phase shift amount

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 4095

Spindle position is saved by CNC. Spindle phase synchronous control whose phase shift amount is spindle position saved to this parameter can be executed.

Alarm and message

Number	Message	Description
SP1256	SPINDLE PHASE-SYNC IMPOSSIBLE	Arbitrary spindle position phase synchronous control cannot be executed. Save the spindle position.

10.15 SPINDLE ORIENTATION

Overview

This function stops the spindle at a specified position. The spindle can be stopped in either of the following two ways.

-The spindle is stopped by applying a function of the spindle control unit.
-The spindle is stopped using mechanical stop.

Explanation

- Using the spindle control unit

Some spindle control units can position the spindle motor by using sensors and position coders. The CNC itself does not control positioning by using these units.

- Mechanical stop

To mechanically stop the spindle by using, for example, a shot pin, rotate the spindle at a constant low speed and drive the pin into the spindle. The spindle can be rotated at a low constant speed by applying either of the following methods.

- Spindle orientation signal SOR<Gn029.5>
- Spindle output control by the PMC

- Serial spindle orientation by a position coder

Two methods of serial interface spindle orientation are available:

- Spindle orientation by a magnetic sensor
- Spindle orientation by a position coder

Spindle orientation by a position coder is described below.

For the function of spindle orientation by a magnetic sensor, refer to "FANUC SPINDLE MOTOR α*i* series PARAMETER MANUAL (B-65280EN)".

Two methods of spindle orientation by a position coder are available:

- Orientation with the fixed stop position
- Orientation with the stop position set externally

(1) Orientation with the fixed stop position: Bit 0 (ORT) of parameter No.3729 = 0
By inputting the spindle orientation command signal, the spindle can always be stopped at an orientation stop position set beforehand with a parameter.

(2) Orientation with the stop position set externally: Bit 0 (ORT) of parameter No.3729 = 1

By externally setting the 12-bit orientation external stop position command signal and inputting the spindle orientation command signal, the spindle can always be stopped at an arbitrary orientation position within one spindle rotation.

- Least input increment

Least input increment of the stop position for spindle orientation can be changed by bit 0 (ORPUNT) of parameter No.4542.
Moreover, data range are changed at the stop position (parameters Nos.4031 and 4204), stop position shift amount (parameters Nos.4077 and 4228), in-position width (parameters Nos.4075 and 4226) and spindle orientation signals with the stop position externally set . Detail is shown in Table 10.15 (a) below.

Table 10.15 (a)

	Bit 0 (ORPUNT) of No.4542	
	0	**1**
Least input increment [deg]	360/4096=0.08789	360/32768=0.01098
Stop position for orientation Data range of parameters Nos.4031,4204	0 to 4095	0 to 32767
Stop position shift amount for orientation Data range of parameters Nos.4077,4228	-4095 to 4095	-32767 to 32767
Detection level for the spindle orientation completion signal Data range of parameters Nos.4075,4226	0 to 100	0 to 1000
Spindle orientation signals with the stop position externally set Data range of signals below SH00A to SH14A<Gn078,Gn079>, SH00B to SH14B<Gn080,Gn081>, SH00C to SH14C<Gn208,Gn209>, SH00D to SH14D<Gn270,Gn271>	0 to 4095	0 to 32767

- Wait time after changing signals with stop position externally set

When bit 0 (ORPUNT) of parameter No.4542 is set to 1, the wait time from the spindle orientation signals with the stop position externally set change to the commanded orientation command signal is reduced. Table 10.15 (b) is shown the wait time.

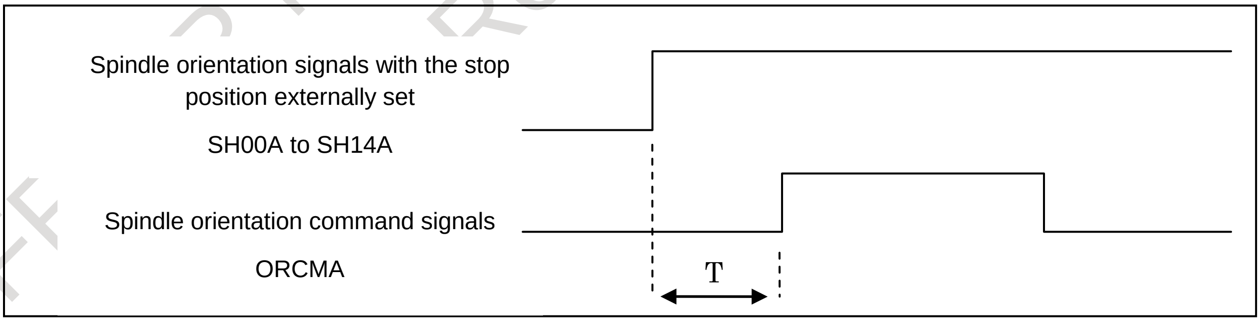


Table 10.15 (b)

	Bit 0 (ORPUNT) of No.4542	
	0	**1**
Wait time T[msec]	50	0

For the function of spindle orientation by a position coder, refer to "FANUC SPINDLE MOTOR α *i* series PARAMETER MANUAL (B-65280EN)", too.

Signal

Spindle orientation signal SOR<Gn029.5>

[Classification] Input signal

[Function] The spindle or the spindle motor is rotated at a constant

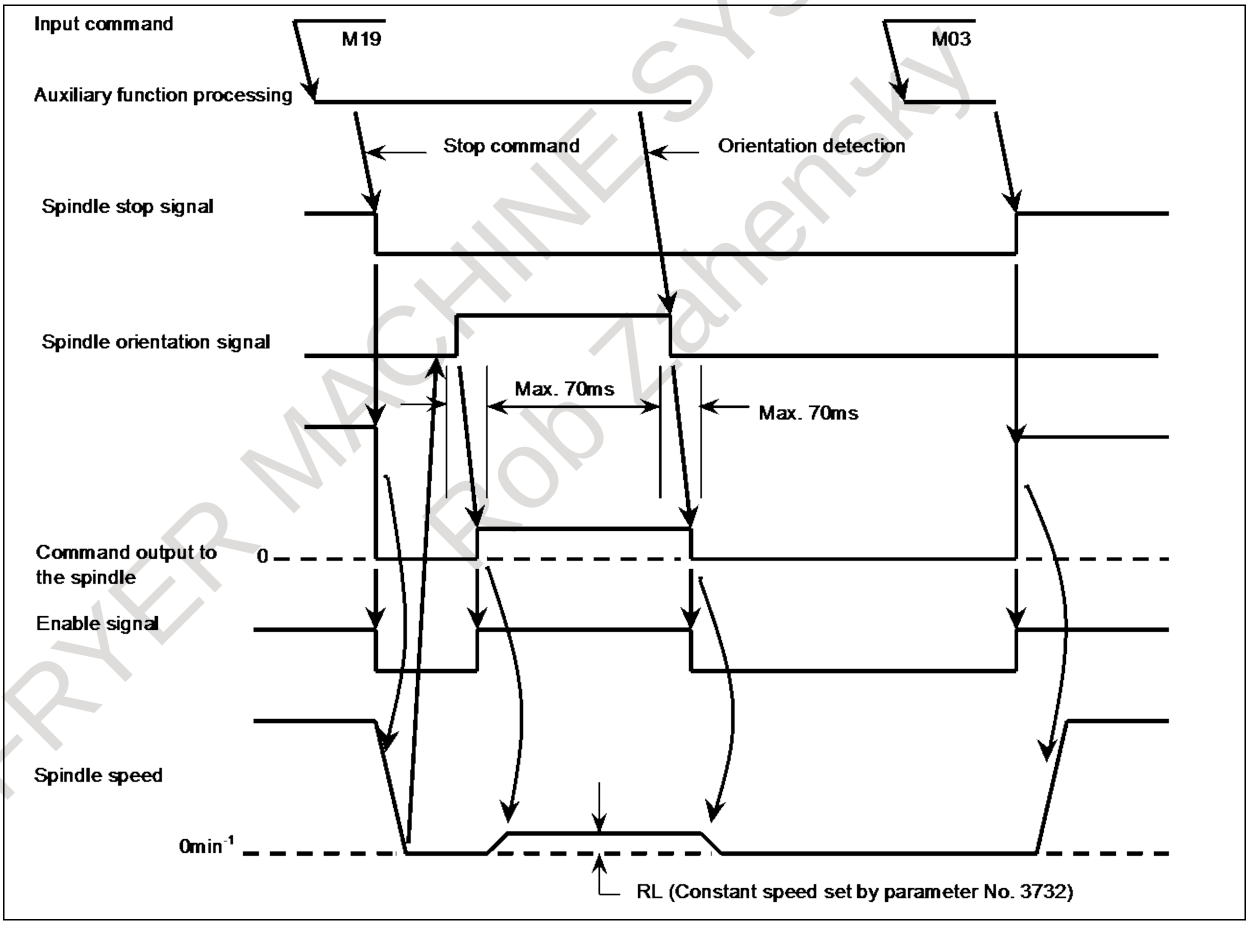
[Operation] When the spindle orientation signal turns to “1” and the spindle stop signal *SSTP turns to “0”, a spindle speed command which lets the spindle rotate at the constant speed set by parameter No. 3732 is output. The enable signal ENB also turns to “1”. This signal is disabled when the spindle stop signal is “1”.

In the machining center system, when the spindle speed for orientation is set by bit 1 (GST) of parameter No. 3705 and the SOR signal is input, the CNC outputs the spindle speed command corresponding to the speed set to parameter 3732 with an output polarity set by bit 5 (ORM) of parameter No. 3706, but the gear select signal does not change.

For example, if the SOR signal is turned to 1 with high gear selected, and the speed set to parameter No. 3732 is in the low gear range, the gear select signal does not change and the command output is calculated and output to obtain the set speed at high gear.

When the spindle motor speed is set by bit 1 (GST) of parameter No. 3705=1, the command output is output regardless of gear select signal. When the spindle motor speed is set, it is used for gear shift.

Example of usage is shown below:



Spindle orientation command signals
ORCMA<Gn070.6> : For first spindle
ORCMB<Gn074.6> : For second spindle
ORCMC<Gn204.6> : For third spindle
ORCMD<Gn266.6> : For fourth spindle

[Classification] Input signal

[Function] These signals specify spindle orientation for serial spindle amplifiers.
For details of the sequence, refer to "FANUC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)".

Spindle orientation signals with the stop position externally set
SH00A to SH14A<Gn078,Gn079> : For first spindle
SH00B to SH14B<Gn080,Gn081> : For second spindle
SH00C to SH14C<Gn208,Gn209> : For third spindle
SH00D to SH14D<Gn270,Gn271> : For fourth spindle

[Classification] Input signal

[Function] This command is used for specifying a stop position with an absolute position within one rotation in the following equation:

- Bit 0(ORPUNT) of parameter No.4542 is set to 0 :
  Effective 12bit (SH00A to SH11A, SH00B to SH11B, SH00C to SH11C, SH00D to SH11D)

$$\textit{Stop position}\ (\mathrm{deg}) = \frac{360}{4096} \times \sum_{i=0}^{11} (2^i \times \mathrm{Pi})$$

- Bit 0(ORPUNT) of parameter No.4542 is set to 1 :
  Effective 15bit (SH00A to SH14A, SH00B to SH14B, SH00C to SH14C, SH00D to SH14D)

$$\textit{Stop position}\ (\mathrm{deg}) = \frac{360}{32768} \times \sum_{i=0}^{14} (2^i \times \mathrm{Pi})$$

where Pi = 0 when SHiA = 0, or Pi = 1 when SHiA = 1.
For details of the sequence, refer to "FANUC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)".

> **NOTE**
> When bit 0 (ORPUNT) of parameter No.4542 is set to 0, after the spindle orientation signals with the stop position externally set is set, change the orientation signals ORCM after waiting for 50msec or more.
> When bit 0 (ORPUNT) of parameter No.4542 is set to 1, the wait time is unnecessary.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn070		ORCMA						
Gn078	SH07A	SH06A	SH05A	SH04A	SH03A	SH02A	SH01A	SH00A
Gn079		SH14A	SH13A	SH12A	SH11A	SH10A	SH09A	SH08A
Gn074		ORCMB						
Gn080	SH07B	SH06B	SH05B	SH04B	SH03B	SH02B	SH01B	SH00B

Gn081		SH14B	SH13B	SH12B	SH11B	SH10B	SH09B	SH08B
Gn204		ORCMC						
Gn208	SH07C	SH06C	SH05C	SH04C	SH03C	SH02C	SH01C	SH00C
Gn209		SH14C	SH13C	SH12C	SH11C	SH10C	SH09C	SH08C
Gn266		ORCMD						
Gn270	SH07D	SH06D	SH05D	SH04D	SH03D	SH02D	SH01D	SH00D
Gn271		SH14D	SH13D	SH12D	SH11D	SH10D	SH09D	SH08D

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3729								ORTs

[Input type] Parameter input
[Data type] Bit spindle

#0 ORTs When a serial spindle is used, the spindle orientation function of stop position external setting type based on the position coder is:
0: Not performed.
1: Performed.

> **NOTE**
> When bit 0 (ORPUNT) of parameter No.4542 is set to 1, if bit 0 (ORTs) of parameter No.3729 is changed, the power must be turned off before operation is continued.

4031	Stop position in orientation by a position coder (MAIN spindle)
4204	Stop position in orientation by a position coder (SUB spindle)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] Depend on the bit 0 (ORPUNT) of parameter No.4542 setting (Refer to the table below)

Bit 0 (ORPUNT) of No.4542	Least input increment	Valid data range
0	360/4096= 0.08789	0 to 4095
1	360/32768= 0.01098	0 to 32767

Each of these parameters specifies a stop position in orientation by a position coder.
These parameters are valid when bit 0 (ORT) of parameter No.3729 is set to 0.

4075	Orientation completion signal detection level (limits of in-position)（MAIN spindle）
4226	Orientation completion signal detection level (limits of in-position)（SUB spindle）

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] Depend on the bit 0 (ORPUNT) of parameter No.4542 setting (Refer to the table below)

Bit 0 (ORPUNT) of No.4542	Valid data range
0	0 to 100
1	0 to 1000

This data is used to set the detecting level of orientation completion signal (ORARA). ORARA is assumed to be 1 if the position error is within the setting.

4077	Stop position shift amount in orientation by a position coder(MAIN spindle)
4228	Stop position shift amount in orientation by a position coder(SUB spindle)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] Depend on the bit 0 (ORPUNT) of parameter No.4542 setting (Refer to the table below)

Bit 0 (ORPUNT) of No.4542	Least input increment	Valid data range
0	360/4096= 0.08789	-4095 to 4095
1	360/32768= 0.01098	-32767 to 32767

Each of these parameters specifies a stop position shift amount in orientation by a position coder.
This parameter is valid no matter what the setting of bit 0 (ORT) of parameter No.3729 is.

	#7	#6	#5	#4	#3	#2	#1	#0
4542								ORPUNT

[Input type] Parameter input
[Data type] Bit spindle

#0 ORPUNT Least input increment for Stop position of spindle orientation, Stop position shift amount, In-position width
0: 360/4096 = 0.08789 [deg]
1: 360/32768 = 0.01098 [deg]

> ⚠ **CAUTION**
> In the α*i* position coder, the effect is not achieved by the fine resolution application of stop position because the sensor resolution is 4096p/rev.

> **NOTE**
> To use this function, the corresponding serial spindle software is required.

Alarm and message

Number	Message	Description
SP1258	ILLEGAL SPINDLE PARAMETER SETTING (SOFT)	The parameter is an illegal setting. Check as follows. - Bit 0 (ORPUNT) of parameter No.4542 is set to 1 the spindle control software that does not support this function. Set 0 to the bit 0 (ORPUNT) of parameter No.4542.

Diagnosis data

445	Position data of position coder (pulse)

[Data type] 2-word spindle
[Unit of data] Pulse
[Valid data range] Depend on the bit 0 (ORPUNT) of parameter No.4542 setting (Refer to the table below)

Bit 0 (ORPUNT) of No.4542	Valid data range
0	0 to 4095
1	0 to 32767

In serial spindle, position coder signal pulse data from one-rotation signal is displayed as the spindle position data. It's effective when bit 1 (SPP) of parameter No.3117 is set to 1. For displaying spindle position data, spindle orientation must be executed once.

449	Position data of position coder (angle)

[Data type] Real spindle
[Unit of data] Refer to the table below
[Valid data range] 0 to 359.999[deg]

Diagnosis data No.449 indicates spindle angle from the one-rotation signal when bit 1 (SPP) of parameter No.3117 is set to 1. Spindle angle displays to three decimal places. It is available for serial spindle. To display spindle angle, spindle orientation must be performed once. Unit of spindle angle data is as follows.

Bit 0 (ORPUNT) of No.4542	Unit of data [deg]
0	0.088
1	0.011

NOTE

Once spindle orientation is performed after power-on, position data can be read. To read position data, set bit 1(SPP) of parameter No. 3117 to 1.
(Except when maintenance is performed, bit 1(SPP) of parameter No. 3117 must be set to 0.)

Caution

⚠ CAUTION

1 To perform spindle orientation by using the spindle control unit, the signals of the spindle control unit must be used.
To perform serial spindle orientation by using a position coder (to perform serial spindle orientation with the stop position set externally), the serial spindle control unit signals must be used.
2 When the function of spindle orientation with the stop position set externally is used, the parameters for stop positions in orientation by a position coder (parameters Nos.4031 and 4204) are invalid.
3 In the αi position coder, the effect is not achieved by the fine resolution application of stop position because the sensor resolution is 4096p/rev.
4 The precision for stop position is depended by the specification of sensor.

Note

NOTE
1. Stop position external setting type orientation cannot be used in Magnetic sensor method spindle orientation.
2. Spindle orientation with the spindle positioning function differs from that described in this section. For details, see "Spindle Positioning."

Reference item

Manual name	Item name
FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)	Position coder method spindle orientation

10.16 SPINDLE OUTPUT SWITCHING

Overview

Spindle output switching switches between the two motor windings, one for low speed and the other for high speed, incorporated into the special spindle motors. This ensures that the spindle motor demonstrates stable output characteristics over a wide range.
Since spindle output switching is a function of the spindle control unit, also refer to the manual for the spindle control unit being used.
This section describes the relationship between spindle output switching and the spindle control function in the CNC.

NOTE
Ladder sequence of the spindle output switching can be easily incorporated by using PMC function. The FANUC LADDER-III CD, function block that can be incorporated easily spindle output switching is stored in the form of a library. For details on library of function block, refer to the documents in the FANUC LADDER-III CD. For details on function block, refer to the PMC PROGRAMMING MANUAL (B-64513EN).

Explanation

- Operation of spindle output-switchable spindle motor

To switch the spindle output characteristics, the windings are usually switched using a relay. Prior to the completion of winding switching, the spindle rotates free from drive power.
Output switching changes the relationship between a speed command, issued from the CNC to the spindle, and the output characteristics of the spindle motor. However, the relationship between the speed command and spindle motor speed is not changed.

- Spindle output switching timing

During actual machining, the spindle is usually controlled in the following way.
(1) Constant spindle speed during cutting, such as milling
(2) Continuously changing spindle speed during cutting, such as in constant surface speed control
(3) Controlling the position loop including the spindle motor during rigid tapping, spindle positioning, Cs contour control, etc.

For applications such as those in (1), we recommend switching the output characteristics for low speed and high speed by using the spindle motor speed detection signal of the spindle control unit.
For applications such as those described in (2) and (3), the spindle must not rotate with no drive power applied during cutting or positioning. It is necessary for the output characteristics to be switched appropriately before machining or for output switching to be masked by using a PMC ladder sequence.

- Output switching and gear switching

Spindle output switching ensures that the spindle motor demonstrates stable characteristics over a wide range, and eliminates the requirement of a mechanical spindle gear switching mechanism.
In creating a PMC ladder sequence for output switching, however, using the gear switching of the CNC's spindle control function may facilitate programming.
Note the following points when using gear switching for CNC spindle control for output switching with a machine tool having no mechanical gear switching mechanism.

- When gear selection output signals, GR1O and GR2O <Fn034.0, 1>, are used (for machining centers in which constant surface speed control is not provided and GTT, bit 4 of parameter No. 3706, is set to 0)
  Set two gears, which are almost the same.
  (Example: Value of parameter No.3741 = value of parameter No.3742 - 1, value of No.3742 = Maximum spindle speed)
  When parameter No.3741 is equal to parameter No.3742, the CNC judges that one gear is used, and does not output the GR2O signal.
- The parameters related to gear switching points, SGT, bit 3 of parameter No.3705, and SGB, bit 2 of parameter No.3705, parameters Nos.3761 and 3751 can be used.
- In usual spindle control, depending on the speed at switching points, the speed specified by the spindle speed command may differ slightly from the actual speed in the area where the maximum spindle speed is set to the maximum speed - 1. (This is because the spindle motor speed, specified by the speed command, is calculated based on the settings of parameters Nos.3741 to 3744.)
  This does not apply to rigid tapping. (Because the machine tool is controlled using the feedback signal from the detector in the position loop.)
- When gear selection input signals, GR1 and GR2 <Gn028.1, 2>, are used (for lathes or machining centers in which constant surface speed control is provided or GTT, bit 4 of parameter No.3706, is set to 1)
  Parameter settings are read according to the input signal information. Unlike the GR2O and GR1O signals, these signals do not require special parameter settings.
  Example)
  When parameters Nos.3741 and 3742 are set to the maximum spindle speed.
  Create a PMC sequence that specifies the following.
  For gear 1, set GR1 and GR2 <Gn028.1, 2> to "0".
  For gear 2, set GR2 <Gn028.2> to "0" and set GR1 <Gn028.1> to "1".
  The PMC must determine the switching timing on the basis of some information.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	Spindle speed control Rigid tapping
FANUC AC SPINDLE MOTOR α*i* series PARAMETER MANUAL (B-65280EN)	Spindle switching control

10.17 SPINDLE COMMAND SYNCHRONOUS CONTROL

Overview

Spindles can be synchronized with each other for control.
Two spindles can be combined, one as a master spindle and the other as a slave spindle, so that Cs contouring control, rigid tapping, spindle positioning, and normal spindle rotation control can also be used with the slave spindle under control by the master spindle.
A master spindle and slave spindle to be placed under spindle command synchronous control can be freely selected from the first to fourth spindles.
Multiple combinations of a master spindle and slave spindle can be placed under spindle command synchronous control.
This function can be used with serial spindles.

Unlike spindle synchronous control, spindle command synchronous control does not guarantee spindle operation synchronization between a master spindle and slave spindle. Moreover, applicable spindle functions differ.
The major differences from spindle synchronous control are indicated below.
For details, see also Section of "SPINDLE SYNCHRONOUS CONTROL".

Function		Spindle command synchronous control	Spindle synchronous control
Combination with other spindle functions	**Spindle rotation control**	∆(Same speed command is used.)	O(Synchronization/phase matching is possible.)
	Cs contour control	O	X
	Rigid tapping	O	X
	Spindle positioning	O	X
Synchronization of multiple combinations		O	O
Synchronization of multiple slave spindles		O	O
Parking function		O	X

Explanation

When the spindle command synchronous control signal ESRSYC<Gn064.6> (ESSYC1-ESSYC4 <Gn264.0>-<Gn264.3> when bit 5 (SSY) of parameter No.3704 is set to 1.) is input from the PMC, the spindle command synchronous control mode is selected.

In spindle command synchronous control, a command for the master spindle also functions as a command for the slave spindle.
The ordinary specification method for a master spindle can be used without modification in the spindle rotation control mode, Cs contouring control mode, rigid tapping mode, and spindle positioning mode.

The parking function can stop each spindle under spindle command synchronous control.

- Configuration of synchronized spindles

In spindle command synchronous control, a spindle that accepts an S command/move command is referred to as a master spindle, and a spindle that ignores a command for itself and performs operation in synchronism with the master spindle is referred to as a slave spindle.

When bit 5 (SSY) of parameter No.3704 is set to 0
The configuration of spindles placed under spindle command synchronous control is as follows:

	Master spindle	**Slave spindle**
Spindle number	First spindle	Second spindle

When bit 5 (SSY) of parameter No.3704 is set to 1
Which slave spindle is placed under spindle command synchronous control with which master spindle is set using parameter No.4821.

	Master spindle	Slave spindle
Spindle number	Second, third, or fourth spindle	First spindle
	First, third, or fourth spindle	Second spindle
	First, second, or fourth spindle	Third spindle
	First, second, or third spindle	Fourth spindle

- When four spindles are involved, two combinations of spindles can be formed for spindle synchronization.
  Example)
    Two combinations are possible: first spindle (master spindle)/second spindle (slave spindle) and third spindle (master spindle)/fourth spindle (slave spindle).

- Multiple slave spindles can be synchronized with one master spindle.
  Example)
    A combination of the first spindle (master spindle)/the second, third, and fourth spindles (slave spindles) is possible.

- A combination that uses a slave spindle as a master spindle is not allowed.

- Operation in each control mode

(1) Spindle rotation control mode
The slave spindle rotates upon the issue of the same command as that used for the master spindle.
The command does not specify the speed of the spindle, instead specifies the ratio of the spindle motor speed to the maximum speed. So, only when the spindle unit configuration of master and slave spindles (parameter settings for a maximum motor speed, motor-spindle gear ratio, and so forth) is the same, the two spindles rotate at about the same speed. (However, the synchronous state is not guaranteed.)
For details of spindle rotation control, see “SPINDLE SPEED CONTROL.”

(2) Cs contour control mode
When the Cs contouring control is used together, Cs contouring control on a slave spindle operating in synchronism with the master spindle is possible. (Control independent of the master spindle is impossible.)
A move command address and position are displayed only for the master spindle.
In reference position return operation, the two spindles are checked for reference position return completion before the reference position return completion signal is turned “1”.
In spindle command synchronous control, switching between the spindle rotation control mode and the Cs contouring control mode is possible.
For details of Cs contouring control, see "Cs CONTOUR CONTROL".

(3) Rigid tapping
When the rigid tapping is selected, the slave spindle supports rigid tapping under the control of a rigid tapping command for the master spindle. (Rigid tapping for the slave spindle cannot be performed independently of the master spindle.)
For details of rigid tapping, see “RIGID TAPPING.”

(4) Spindle positioning
When the spindle positioning is selected, the spindle positioning function for the slave spindle is enabled under the control of a spindle positioning command for the master spindle. (The spindle positioning function for the slave spindle cannot be used independently of the master spindle.)

In the same way as with Cs contour axis control, the display of command address and position data is limited to the master spindle. Operations such as orientation are performed in the same way as usual. However, both spindles are checked for the completion of orientation, and the spindle positioning sequence is processed only after the completion of positioning by both spindles has been confirmed.
For details of spindle positioning, see “SPINDLE POSITIONING.”

- Parking function

The parking function stops the rotation and movement of each of master and slave spindles placed in the spindle command synchronous control mode, regardless of the spindle control mode (spindle rotation control, Cs contouring control, rigid tapping, or spindle positioning).
In the parking state, the spindle is placed in the following state, depending on its mode:

Spindle rotation control mode:
Rotation is stopped in the same way as when S0 is specified.
Other modes:
Movement is stopped in the same way as in the case of no distribution pulse.

With a move command for positioning of a Cs contouring control axis or spindle in the parking state, only the movement of the spindle in the parking state is stopped by updating the coordinate system.

Reference position return operation for a Cs contouring control axis and spindle orientation operation for spindle positioning are disabled for a spindle placed in the parking state.
If reference position return operation in the Cs contouring control mode or spindle orientation in the spindle positioning mode is specified with one spindle placed in the parking state, the reference position return completion signal is turned “1” when the spindle not placed in the parking state has completed a reference position return operation.

The parking function stops the movement of only a spindle that is placed in the parking state. So, the parking function can be used, for example, to continue machining by stopping one spindle no longer required or to perform Cs contouring control, rigid tapping, and spindle positioning with the slave spindle by placing the master spindle in the parking state.

NOTE

1 In the spindle command synchronous control mode, the two spindles are checked for a position error at all times, regardless of the parking state. The excessive error alarm (SV0410 or others) is issued when a spindle has moved due to a load from the position where the spindle was stopped by the parking function.
2 The parking function stops the movement of only a spindle that is placed in the parking state. So, if parking is applied to a spindle after a reference position return operation or spindle orientation operation is once performed, the spindle position is shifted from the machine coordinates. Until another reference position return operation or spindle orientation operation is performed after parking is canceled, the relationship between the spindle position and machine coordinates is not guaranteed.
3 The parking signal becomes immediately valid in the spindle command synchronous control mode. However, even if the parking signal is input during reference position return operation in the Cs contouring control mode or during spindle orientation operation in the spindle positioning mode, the reference position return operation or spindle orientation operation continues until it is completed. Parking starts immediately after completion of such an operation.

> **NOTE**
>
> 4 Do not specify a reference position return operation in the Cs contouring control mode or a spindle orientation operation in the spindle positioning mode while the two spindles are placed in the parking state. When the two spindles are placed in the parking state, the reference position return operation cannot be completed.
>
> 5 When using parking signal PK7 or PK8 for spindle synchronous control while both spindle command synchronous control and spindle synchronous control are being used, set the SPK bit (bit 7 of parameter No.4800) to 1. This sets parking signals PKESS1 and PKESS2, used for spindle command synchronous control, to bits 6 and 7 of G031.

- Simple spindle synchronization and phase error monitor signal

Spindle command synchronous control does not guarantee synchronous spindle operation. In spindle control involving a position loop as in the case of Cs contouring control, rigid tapping, and spindle positioning, however, the synchronous state of the two spindles can be maintained by matching the position gains of the two spindles.

For monitoring of the synchronous state of two spindles in such a case, the phase error monitor signal SYCAL<Fn044.4> is available. The error between two spindles is monitored at all times. When a synchronization error exceeding the value set in parameter No.4826 is detected, this signal is set to “1”. When a synchronization error not exceeding the value set in parameter No.4826 is detected, this signal is set to 0.

In the spindle rotation control mode, however, synchronous state monitoring is not performed. So, the phase error monitor signal has no effect, and is set to 0 at all times.

Note that even if phase error monitor signal SYCAL is output, no CNC alarm is issued. This signal is designed to be used by the PMC to monitor the machine synchronization state. Its usage varies depending on the machine tool builder. The use of this signal is not mandatory.

> **NOTE**
>
> During spindle command synchronous control, a reference position return operation in the Cs contouring control mode, a spindle orientation operation in the spindle positioning mode, or a function (such as spindle orientation) that is directly specified for a spindle with a PMC signal is performed for each spindle independently, so that synchronization is not guaranteed.

- Relationship between spindle command synchronous control and spindle synchronous control

The spindle synchronous control mode and the spindle command synchronous control mode cannot be selected for the same spindle at the same time.
In a mode other than spindle synchronous control mode, no restriction is imposed on the selection timing of spindle command synchronous control. Usually, select spindle command synchronous control when the master spindle is not executing a move command in the spindle rotation control mode or in the Cs contouring control mode.
No restriction is imposed on the timing of spindle command synchronous control cancellation. Usually, cancel spindle command synchronous control when no command is executed in any spindle control mode. When spindle command synchronous control is canceled, the mode of the slave spindle switches to the spindle rotation control mode. The master spindle maintains the selected spindle control mode.

The table below indicates the spindle control mode switching of the master and slave spindles based on the transitions of the spindle command synchronous control signal and spindle synchronous control signal.

Spindle control mode switching	Switching when the spindle command synchronous control signal makes a transition from “0” to “1” in each spindle control mode selection state			
Master spindle	SP → SP(ESY)	CT → CT(ESY)	SV → SV(ESY) Note 3	SY → SY Note 4
Slave spindle	SP → SP(ESY) Note 1	SP → CT(ESY) Note 2	SP → SV(ESY) Note 3	SY → SY Note 4

Spindle control mode switching	Switching when the spindle command synchronous control signal makes a transition from “1” to “0” in each spindle command synchronous control mode selection state		
Master spindle	SP(ESY) → SP	CT(ESY) → CT	SV(ESY) → SV
Slave spindle	SP(ESY) → SP	CT(ESY) → SP	SV(ESY) → SP

Spindle control mode switching	Switching when the spindle synchronous control signal makes a transition from “0” to “1” in each spindle command synchronous control mode selection state		
Master spindle	SP(ESY) → SP(ESY) Note 4	CT(ESY) → CT(ESY) Note 4	SV(ESY) → SV(ESY) Note 4
Slave spindle	SP(ESY) → SP(ESY) Note 4	CT(ESY) → CT(ESY) Note 4	SV(ESY) → SV(ESY) Note 4

The symbols used in the tables above have the following meanings:

SP : Spindle rotation control mode
CT : Cs contour control mode
SV : Rigid tapping mode or spindle positioning mode
SP(ESY) : Spindle rotation control mode during spindle command synchronous control
CT(ESY) : Cs contouring control mode during spindle command synchronous control
SV(ESY) : Rigid tapping or spindle positioning mode during spindle command synchronous control
SY : Spindle synchronous control mode

NOTE

1. After initialization to the spindle rotation control mode, operation is performed with a speed command for the master spindle. So, a speed change can occur.
2. Even if the master spindle is already at the reference position, the slave spindle is not at the reference position yet when the mode of the slave spindle is just switched to the Cs contouring control mode by spindle command synchronous control. So, perform a reference position return operation again.
3. When the master spindle is placed in the rigid tapping or spindle positioning mode, the input of the spindle command synchronous control signal is masked in the CNC, so that the mode is not switched to the spindle command synchronous control mode. After the rigid tapping or spindle positioning mode of the master spindle is canceled, the master spindle and slave spindle are placed in the spindle command synchronous control mode. After the spindle command synchronous control mode is once set, the specification or cancellation of the rigid tapping or spindle positioning mode of the master spindle is applied to both of the master spindle and slave spindle.
4. The spindle control mode currently selected is maintained. As an alarm indicating that the spindle control mode is not switched, alarm PS0194, “SPINDLE COMMAND IN SYNCHRO-MODE” is displayed. This alarm can be cleared by resetting the signal input later to “0”.

> **NOTE**
> 5 Switch to the spindle command synchronous control mode when the slave spindle is placed in the spindle rotation control mode.
> 6 For the slave spindle placed in the spindle command synchronous control mode, the switching to a spindle mode such as Cs contouring control, rigid tapping, spindle positioning, and spindle synchronous control cannot be specified.

- Spindle rotation direction

In the spindle command synchronous control mode, the same speed command and move command are issued to the master spindle and slave spindle.
The specified rotation direction of a spindle can be changed using a PMC signal (spindle forward/reverse rotation command signal SFR or SRV) or serial spindle parameter according to the control mode.

- Position deviation check and alarm display

When spindle command synchronous control is used in a mode other than the spindle rotation control mode, the master spindle and slave spindle are checked for a position deviation.
An in-position check is made to see if the position deviation of each of the two spindles is within a parameter-set range.
Position deviation excessive error checks at stop time and move time are made to see if the position deviation of each spindle exceeds a parameter-set value.
The parameters set for the master spindle are used for both spindles.
An alarm for an excessive error is displayed for the master spindle. Which of the master spindle and slave spindle has entered the excessive error state cannot be identified on the screen.

Limitation

- Both of the master spindle and slave spindle must have hardware such as a spindle-related detector required for the functions (Cs contouring control, rigid tapping, and spindle positioning) that use spindle command synchronous control.

Diagnosis data

418	Position deviation of each spindle

When spindles are involved in a position loop, the position deviation of each spindle is indicated.
The unit is pulses. The detection unit per pulse varies from one spindle control mode to another.

425	Synchronization error of each spindle

The absolute value of the synchronization error of each spindle in spindle command synchronous control is indicated.
A synchronization error is indicated with a slave spindle.
In the spindle rotation control mode, the synchronization error is 0 at all times.

Spindle tuning screen

In the spindle command synchronous control mode, the display dedicated to the spindle command synchronous control mode is not provided. Instead, a spindle control mode selected in the spindle command synchronous control mode is displayed.
When the Cs contouring control mode is set in spindle command synchronous control, the Cs contouring control mode is displayed.
For details, refer to Spindle tuning screen of MAINTENANCE MANUAL (B-64695EN).

Signal

Spindle command synchronous control signal ESRSYC<Gn064.6>

[Classification] Input signal

[Function] This signal specifies switching to the spindle command synchronous control mode for the first/second spindles.

[Operation] When this signal is set to “1”, the spindle command synchronous control mode for the first/second spindles is set.
When this signal is set to “0”, the spindle command synchronous control mode for the first/second spindles is canceled.
This signal is valid only when bit 5 (SSY) of parameter No.3704 is set to 0.

Spindle command synchronous control signal for each spindle
ESSYC1<Gn264.0> : For first spindle
ESSYC2<Gn264.1> : For second spindle
ESSYC3<Gn264.2> : For third spindle
ESSYC4<Gn264.3> : For fourth spindle

[Classification] Input signal

[Function] These signals specify switching to the spindle command synchronous control mode for each spindle.

[Operation] When each of these signals is set to “1”, the spindle command synchronous control mode with the corresponding spindle set as a slave spindle is set.
When each of these signals is set to “0”, the spindle command synchronous control mode with the corresponding spindle set as a slave spindle is canceled.
When ESSYC1 is set to “1”: Spindle command synchronous control with the first spindle set as a slave spindle is exercised.
When ESSYC2 is set to “1”: Spindle command synchronous control with the second spindle set as a slave spindle is exercised.
When ESSYC3 is set to “1”: Spindle command synchronous control with the third spindle set as a slave spindle is exercised.
When ESSYC4 is set to “1”: Spindle command synchronous control with the fourth spindle set as a slave spindle is exercised.
These signals are valid only when bit 5 (SSY) of parameter No.3704 is set to 1.
Which spindle is synchronized with which spindle is set using parameter No.4821.

First spindle command synchronous parking signal
PKESS1<Gn122.6> or PKESS1<Gn031.6>

[Classification] Input signal

[Function] This signal parks the first spindle in spindle command synchronous control.

[Operation] When this signal is set to “1”, the first spindle is placed in the parking state.
When this signal is set to “0”, the parking state of the first spindle is canceled.
When bit 7 (SPK) of parameter No.4800 is set to 1, <Gn031.6> is used.
This signal is valid only when bit 5 (SSY) of parameter No.3704 is set to 0.

Second spindle command synchronous parking signal
PKESS2<Gn122.7> or PKESS2<Gn031.7>

[Classification] Input signal

[Function] This signal parks the second spindle in spindle command synchronous control.

[Operation] When this signal is set to “1”, the second spindle is placed in the parking state.
When this signal is set to “0”, the parking state of the second spindle is canceled.
When bit 7 (SPK) of parameter No.4800 is set to 1, <Gn031.7> is used.
This signal is valid only when bit 5 (SSY) of parameter No.3704 is set to 0.

Spindle command synchronous parking signal for each spindle
PKESE1<Gn265.0> : For first spindle
PKESE2<Gn265.1> : For second spindle
PKESE3<Gn265.2> : For third spindle
PKESE4<Gn265.3> : For fourth spindle

[Classification] Input signal

[Function] These signals park each spindle in spindle command synchronous control.

[Operation] When each of these signals is set to “1”, the corresponding spindle is placed in the parking state.
When each of these signals is set to “0”, the parking state of the corresponding spindle is canceled.
When PKESE1 is set to “1”: The first spindle is placed in the parking state.
When PKESE2 is set to “1”: The second spindle is placed in the parking state.
When PKESE3 is set to “1”: The third spindle is placed in the parking state.
When PKESE4 is set to “1”: The fourth spindle is placed in the parking state.
This signal is valid only when bit 5 (SSY) of parameter No.3704 is set to 0.

Phase error monitor signal SYCAL<Fn044.4>

[Classification] Output signal

[Function] This signal posts that in the spindle command synchronous control mode for the first/second spindles, the error pulse difference between the two spindles is greater than a parameter-set value.

[Output cond.] This signal is set to “1” in the following case:
- In the spindle command synchronous control mode, the error pulse difference between two spindles is greater than the value set in parameter No.4826 after spindle command synchronous control is completed.

This signal is set to “0” in the following cases:
- The spindle command synchronous control mode is not set.
- In the spindle command synchronous control mode, the error pulse difference between two spindles is equal to or less than the value set in parameter No.4826.

This signal is valid only when bit 5 (SSY) of parameter No.3704 is set to 0.

Phase error monitor signal for each spindle
SYCAL1<Fn043.0> : For first spindle
SYCAL2<Fn043.1> : For second spindle
SYCAL3<Fn043.2> : For third spindle
SYCAL4<Fn043.3> : For fourth spindle

[Classification] Output signal

[Function] These signals post that in the spindle command synchronous control mode with each spindle set as a slave spindle, the error pulse difference between two spindles is greater than a parameter-set value.

[Output cond.] Each of these signals is set to “1” in the following case:
- In the spindle command synchronous control mode, the error pulse difference between two spindles is greater than the value set in parameter No.4826 after spindle command synchronous control is completed.

Each of these signals is set to “0” in the following cases:
- The spindle command synchronous control mode is not set.
- In the spindle command synchronous control mode, the error pulse difference between two spindles is equal to or less than the value set in parameter No.4826.

These signals are valid only when bit 5 (SSY) of parameter No.3704 is set to 1.

Even in the spindle command synchronous control mode, the signals that need to be input/output or are used to control those functions (such as spindle orientation) directly specified from the PMC for the spindle control unit are provided independently for each spindle.
For first spindle : (DI:G0070 to G0073, DO:F0045 to F0048)
For second spindle: (DI:G0074 to G0077, DO:F0049 to F0052)
For third spindle : (DI:G0204 to G0207, DO:F0168 to F0171)
For fourth spindle : (DI:G0266 to G0269, DO:F0266 to F0269)
When using the spindle command synchronous control function, exercise control on the spindle command synchronous control signals and also manipulate the signals for each spindle as required.
(Such a signal state that the master spindle and slave spindle under spindle command synchronous control can make the same movement needs to be set.)

> **NOTE**
> In the spindle command synchronous control mode, the states of both the master spindle and slave spindle are checked. So, if spindle command synchronous control is specified in the Cs contouring control mode before the slave spindle is not activated yet, for example, the VRDY OFF alarm may be issued. Pay attention to control on the PMC signals for the slave spindle as well.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn031	PKESS2	PKESS1						
Gn064		ESRSYC						
Gn122	PKESS2	PKESS1						
Gn264					ESSYC4	ESSYC3	ESSYC2	ESSYC1
Gn265					PKESE4	PKESE3	PKESE2	PKESE1
Fn043					SYCAL4	SYCAL3	SYCAL2	SYCAL1
Fn044				SYCAL				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3704			SSY					

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#5 SSY Simple synchronous spindle control by each spindle is:
0: Not performed.
1: Performed.
The master axis and slave axis of simple synchronous spindle control can be selected from the arbitrary spindles.

The target spindle of simple synchronous spindle control is set in parameter No. 4821.
In addition, the following signals affect the control.

- Signals of simple synchronous control of each spindle ESSYCs
- Parking signals of simple synchronous control of each spindle PKESEs

	#7	#6	#5	#4	#3	#2	#1	#0
4800	SPK	EPZ						

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#6 EPZ When the parking signal is switched in the reference position established state during Cs contour control exercised using spindle command synchronous control:
0: Reference position established state is continued.
1: Reference position established state is canceled.
If this parameter is set, the same reference position return operation as manual reference position return is performed with the G28 command immediately after the parking signal is switched.
The G00 command performs a positioning operation including reference position return (when bit 1 (NRF) of parameter No.3700 is set to 0).

#7 SPK As the parking signals for spindle command synchronous control:
0: PKESS1<Gn122.6> (first spindle) and PKESS2<Gn122.7> (second spindle) are used.
1: PKESS1<Gn031.6> (first spindle) and PKESS2<Gn031.7> (second spindle) are used.

> **NOTE**
> 1 This parameter is valid only when bit 5 (SSY) of parameter No. 3704 is set to 0.
> 2 If the parking signals PK7 and PK8 for synchronization control are used when spindle command synchronous control and synchronization control are used at the same time, set bit 7 (SPK) of parameter No.4800 to 1 to use the parking signals PKESS1 and PKESS2 for spindle command synchronous control as <Gn031.6,Gn031.7>.

	#7	#6	#5	#4	#3	#2	#1	#0
3716								A/Ss

[Input type] Parameter input
[Data type] Bit spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 A/Ss Spindle motor type is :
0: Analog spindle.

1: Serial spindle.

3717	Spindle amplifier number to each spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled axes

Set a spindle amplifier number to be assigned to each spindle.

0: No spindle amplifier is connected.
1: Spindle motor connected to amplifier number 1 is used.
2: Spindle motor connected to amplifier number 2 is used.
to
n: Spindle motor connected to amplifier number n is used.

4821	Master axis of each slave spindle under simple synchronous spindle control

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled spindle axes (within a path)

When a spindle is set as a slave spindle in spindle command synchronous control on each spindle, set which spindle (master spindle) the slave spindle is to be synchronized with.

[Examples of parameter setting]

- When spindle command synchronous control is exercised with the first spindle selected as a master spindle and the second spindle selected as a slave spindle:
  No.4821(1)=0
  No.4821(2)=1
  No.4821(3)=0
  No.4821(4)=0

- When spindle command synchronous control is exercised with four spindles under the following combinations:
  (Two combinations, namely, first spindle (master spindle)/ second spindle (slave spindle), and third spindle (master spindle)/fourth spindle (slave spindle))
  No.4821(1)=0
  No.4821(2)=1
  No.4821(3)=0
  No.4821(4)=3

NOTE
1 This parameter is valid if bit 5 (SSY) of parameter No.3704 is set to 1.
2 The setting of a slave spindle as a master spindle is invalid. Be sure to set 0 for a spindle that is to function as a master spindle.
3 In this parameter, set a spindle number within the same path.

4826	Allowable error count for the error pulses between two spindles in the simple synchronization spindle control mode

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 32767

Set the allowable error count for the error pulses between two spindles in the simple synchronization spindle control mode.
This parameter is used to check a spindle synchronization error phase difference.
When a spindle synchronization error equal to or greater than the value set in this parameter is detected, the spindle phase error monitor signals SYCAL<Fn044.4> and SYCALs are set to “1”.

NOTE
1 The detection unit per pulse depends on the spindle control mode (Cs contour control, rigid tapping, or spindle positioning).
2 Set this parameter for a spindle that is to function as a slave spindle. Set 0 for the master spindle.
3 In the spindle rotation control mode, synchronization error detection is not performed.

Alarm and message

Number	Message	Description
PS0194	SPINDLE COMMAND IN SYNCHRO-MODE	A Cs contour control mode, spindle positioning command, or rigid tapping mode was specified during the spindle synchronous control mode or spindle command synchronous control mode.

Caution

⚠ **CAUTION**
1 As an output signal for monitoring a phase error during spindle command synchronous control, the phase error monitor signal SYCAL is available. The action to be taken when a phase error is detected depends on the specifications of the machine tool builder.
2 In multi-path control, spindle command synchronous control cannot be exercised on spindles each belonging to a different path.

Reference item

For details of the parameters Nos.4000 to 4539, signals, and alarms for the serial spindle control unit, refer to the manuals related to serial spindles listed below.
FANUC AC SPINDLE MOTOR α*i* series PARAMETER MANUAL (B-65280EN)

10.18 SPINDLE COMMAND SYNCHRONOUS CONTROL INDEPENDENT PITCH ERROR COMPENSATION FUNCTION

Overview

When Cs contouring control axes are placed under spindle command synchronous control, pitch error compensation can be exercised on the master spindle and slave spindle independently of each other.

With the bi-directional pitch error compensation option, bi-directional pitch error compensation can also be enabled.

Explanation

When the master spindle is exercising Cs contouring control in spindle command synchronous control, the slave spindle is also placed in the Cs contouring control mode.
An address and position related to a move command are displayed only for the master spindle, and the slave spindle performs the same operation as specified by the move command for the master spindle. The same pitch error compensation as applied to the master spindle is applied to the slave spindle.
The same timing and compensation amount of pitch error compensation are used for the master spindle and slave spindle.

A pitch error compensation amount different from that for the master spindle can be applied to the slave spindle by setting bit 1 (EPC) of parameter No.3601 to 1 and setting the parameters for the slave spindle (pitch error compensation reference position compensation number, minus side compensation point number, and plus side compensation point number) and pitch error compensation data for the slave spindle.
The same timing of pitch error compensation application is applied to the master spindle and slave spindle.
The parameter settings of a pitch error compensation interval and compensation magnification for the master spindle are valid.
Pitch error compensation point numbers specific to the slave spindle are required in addition to those used for the master spindle.

When bi-directional pitch error compensation is used, a different type of pitch error compensation can be applied to a movement in each of the positive and negative directions.

- Example of setting spindle command synchronous control independent pitch error compensation)

- Parameter setting

Parameter number	Description	Setting value
3601#1	Enable independent pitch error compensation for Cs contouring control axes	1
3620	Reference position compensation number (master spindle)	60
3621	Minus side compensation point number (master spindle)	61
3622	Plus side compensation point number (master spindle)	68
3623	Compensation magnification (common)	2
3624	Compensation point interval (common)	45000
3661 (slave spindle number)	Reference position compensation number (slave spindle)	69
3666 (slave spindle number)	Minus side compensation point number (slave spindle)	70
3671 (slave spindle number)	Plus side compensation point number (slave spindle)	77

- Pitch error compensation data (master spindle side)

Compensation number	60	61	62	63	64	65	66	67	68
Setting value	+1	-2	+1	+3	-1	-1	-3	+2	+1
Compensation value	+2	-4	+2	+6	-2	-2	-6	+4	+2

- Pitch error compensation data (slave spindle side)

Compensation number	69	70	71	72	73	74	75	76	77
Setting value	+3	+2	+4	-3	-4	-6	+4	0	+3
Compensation value	+6	+4	+8	-6	-8	-12	+8	0	+6

Suppose that the settings above are made. When the master spindle under spindle command synchronous control outputs the compensation pulses (+2) of compensation number 60, for example, the compensation pulses (+6) of compensation number 69 are output to the slave spindle.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3601							EPC	

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 EPC The pitch error compensation on an axis of Cs contour control on the slave spindle side during simple synchronous spindle control is:
0: The same as that on the master spindle.
1: Just for the slave spindle.

3661	Number of a pitch error compensation position for the reference position for each slave axis when independent pitch error compensation is performed under spindle command synchronous control

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 1535
Set the number of a pitch error compensation position for the reference position.

> **NOTE**
> 1 This parameter is valid if pitch error compensation on an axis of Cs contour control on the salve side during simple synchronous spindle control is carried out just for the slave axis (bit 1 (EPC) of parameter No.3601 is set to 1).
> 2 The usable number of pitch error compensation positions and their range depend on the option configuration.

3666	Number of the pitch error compensation position at extremely negative position for each slave axis when independent pitch error compensation is performed under spindle command synchronous control

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 1535

Set the compensation position number at the farthest end in the negative direction.

NOTE
1. This parameter is valid if pitch error compensation on an axis of Cs contour control on the salve side during simple synchronous spindle control is carried out just for the slave axis (bit 1 (EPC) of parameter No.3601 is set to 1).
2. When using the both-direction pitch error compensation function, set a compensation position number for a movement in the positive direction.
3. The usable number of pitch error compensation positions and their range depend on the option configuration.

3671	Number of the pitch error compensation position at extremely positive position for each slave axis when independent pitch error compensation is performed under spindle command synchronous control

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 1535

Set the compensation position number at the farthest end in the positive direction.

NOTE
1. This parameter is valid if pitch error compensation on an axis of Cs contour control on the salve side during simple synchronous spindle control is carried out just for the slave axis (bit 1 (EPC) of parameter No.3601 is set to 1).
2. When using the both-direction pitch error compensation function, set a compensation position number for a movement in the positive direction.
3. The usable number of pitch error compensation positions and their range depend on the option configuration.

3676	Number of the pitch error compensation position at extremely negative position for each slave axis when independent both-direction pitch error compensation is performed under spindle command synchronous control

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 1535

When using both-direction pitch error compensation, set the compensation position number at the farthest end in the negative direction for a movement in the negative direction.

NOTE
1 This parameter is valid if pitch error compensation on an axis of Cs contour control on the salve side during simple synchronous spindle control is carried out just for the slave axis (bit 1 (EPC) of parameter No.3601 is set to 1).
2 The usable number of pitch error compensation positions and their range depend on the option configuration.

3681	Pitch error compensation value at the reference position when a movement is made to the reference position in the direction opposite to the reference position return direction for each slave axis in the case where independent both-direction pitch error compensation is performed under spindle command synchronous control

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] -32767 to 32767

By using an absolute value, set a pitch error compensation value at the reference position when a movement is made in the negative direction if the reference position return direction (bit 5 (ZMI) of parameter No.1006) is positive or when a movement is made in the positive direction if the reference position return direction (bit 5 (ZMI) of parameter No.1006) is negative.

NOTE
This parameter is valid if pitch error compensation on an axis of Cs contour control on the salve side during simple synchronous spindle control is carried out just for the slave axis (bit 1 (EPC) of parameter No.3601 is set to 1).

Caution

Usable numbers and ranges of pitch error compensation points

Option configuration	Number of pitch error compensation points	Pitch error compensation range
Stored type pitch error compensation only	1536	0 to 1535

Option configuration	Number of pitch error compensation points	Pitch error compensation range
Stored type pitch error compensation + bi-directional pitch error compensation	3072	0 to 1535 3000 to 4535 (Note)

NOTE
Such a setting that the data of one axis extends beyond a pitch error compensation range is prohibited.

Example) A setting that extends from 1535 to 3000 is prohibited.

10.19 SPINDLE SPEED FLUCTUATION DETECTION

Overview

With this function, an overheat alarm OH0704, "OVERHEAT:SPINDLE" is raised and the spindle speed fluctuation detection alarm signal SPAL is issued when the spindle speed deviates from the specified speed due to machine conditions.
This function is useful, for example, for preventing the seizure of the guide bushing.
G26 enables spindle speed fluctuation detection.
G25 disables spindle speed fluctuation detection.

Explanation

The function for detecting spindle speed fluctuation checks whether the actual speed varies for the specified speed or not. Si or Sr, whichever is greater, is taken as the allowable fluctuation speed (Sm). An alarm OH0704 "OVERHEAT:SPINDLE" is activated when the actual spindle speed varies for the commanded speed (Sc) under the condition that the variation width exceeds the allowable variation width (Sm).

$|Sc - Sa| > Sm$

Sc : Specified spindle speed
Sa : Actual spindle speed
Si : The allowable constant variation width which is independent of the specified spindle speed (parameter No.4913)
Sr : The allowable variation width which is obtained by multiplying Sc (commanded spindle speed) by r (constant ratio) (r = parameter No.4912)

Bit 0 (FLR) of parameter No. 4900 = 0	Bit 0 (FLR) of parameter No. 4900 = 1
$Sr = Sc \times \frac{r}{100}$	$Sr = Sc \times \frac{r}{1000}$

Sm: Si or Sr, whichever is greater

- Conditions to start spindle speed fluctuation detection

If the specified spindle speed Sc changes, spindle speed fluctuation detection starts when one of the conditions below is met.
If bit 7 (FDTs) of parameter No.4900 is 1, spindle speed fluctuation detection starts when condition (2) is met.

(1) The actual spindle speed falls in a range of (Sc - Sq) to (Sc + Sq)
Sc : Specified spindle speed
Sq : Tolerance within which the spindle is assumed to attain the programmed speed (parameter No.4911)

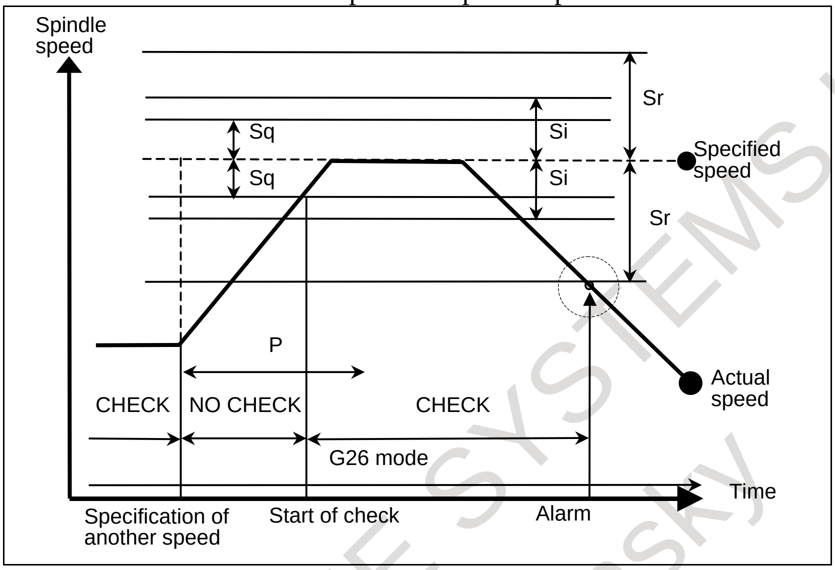
Bit 0 (FLR) of parameter No. 4900= 0	Bit 0 (FLR) of parameter No. 4900= 1
$Sq = Sc \times \frac{q}{100}$	$Sq = Sc \times \frac{q}{1000}$

(2) When time p specified in parameter No.4914 elapses after the specified speed Sc changes.

- Examples of spindle speed fluctuation detection

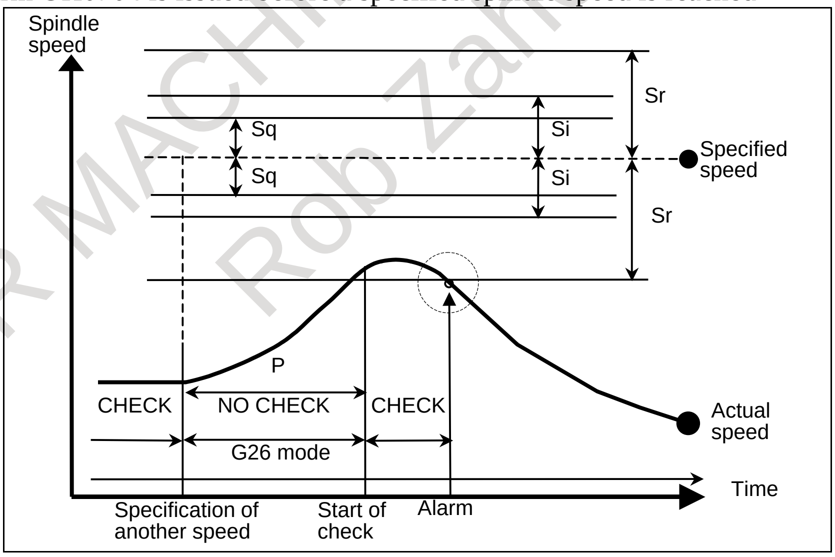
(Example 1)

When an alarm OH0704 is issued after a specified spindle speed is reached



(Example 2)

When an alarm OH0704 is issued before a specified spindle speed is reached



Specified speed : (Speed specified by address S and five-digit value) × (spindle override)

Actual speed : Speed detected with a position coder

p : Period after a change occurs in the actual spindle speed until detection starts
Parameter No.4914, address P

Sq : (Specified spindle speed) × (Detection start tolerance (q))
Parameter No.4911, address Q

Bit 0 (FLR) of parameter No. 4900 = 0	Bit 0 (FLR) of parameter No. 4900 = 1
$\frac{q}{100}$	$\frac{q}{1000}$

Sr : (Specified spindle speed) × (Allowable variation (r))
Parameter No.4912, address R

Bit 0 (FLR) of parameter No. 4900 = 0	Bit 0 (FLR) of parameter No. 4900 = 1
$\frac{r}{100}$	$\frac{r}{1000}$

Si : Allowable variation width
Parameter No.4913, address I

If the difference between the specified speed and actual speed exceeds both Sr and Si, an alarm OH0704 is raised.

- Relationship between spindle speed control and each spindle

Spindle / Function	Serial spindle			
	1st spindle	2nd spindle	3rd spindle	4th spindle
Spindle speed fluctuation detection	Possible	Possible(*1)	Possible(*1)	Possible(*1)

Spindle / Function	Analog spindle			
	1st spindle			
Spindle speed fluctuation detection	Possible			

NOTE

1 Please enable the multi spindle control (bit 3 (MSP) of parameter No.8133 is 1).

2 The spindle speed fluctuation detection function is effective for a single spindle. The function cannot be executed for two or more spindles.
The spindle speed fluctuation detection function is effective for a spindle on which the currently selected position coder is mounted. Just a single position coder can be selected. Multiple position coders cannot be selected. For the selection of a position coder, see the section of "Multi-spindle control."
* Position coder selection signals (PC2SLC<Gn028.7>, PC3SLC<Gn026.0>, PC4SLC<Gn026.1>)

3 The parameters that become valid are the parameters of the spindle speed fluctuation detection function (No.4911, No.4912, No.4913, No.4914) for the spindle on which the currently selected position coder is mounted.

- Spindle that can be subjected to the spindle speed fluctuation detection function, depending on the relationship between system configuration and spindle

The table below show some examples explaining the spindle to be subjected to the spindle speed fluctuation detection function, depending on the system configuration and spindle assignment.

NOTE

1 If path control is performed, the spindle speed fluctuation detection function is carried out for the axis on which the currently selected position coder is mounted.
* Position coder selection signals (PC2SLC<Gn028.7>, PC3SLC<Gn026.0>, PC4SLC<Gn026.1>)

2 Under multi-path control, if the feedback pulse of a position coder mounted on a spindle belonging to a path is used, the selected path feedback must be considered in addition to the selection of the position coder. Great attention should be given to the selection statuses of relative signals.
* Path spindle feedback selection signals (SLPCA<Gn064.2>, SLPCB<Gn064.3>, SLPCC<Gn403.4>, SLPCD<Gn403.5>)

> **NOTE**
> 3 It is assumed that the reader has understood the individual functions described in this manual. If necessary, see the sections of "spindle control for each path," "spindle control," and "multi-spindle control."

Examples of spindles subjected to detection

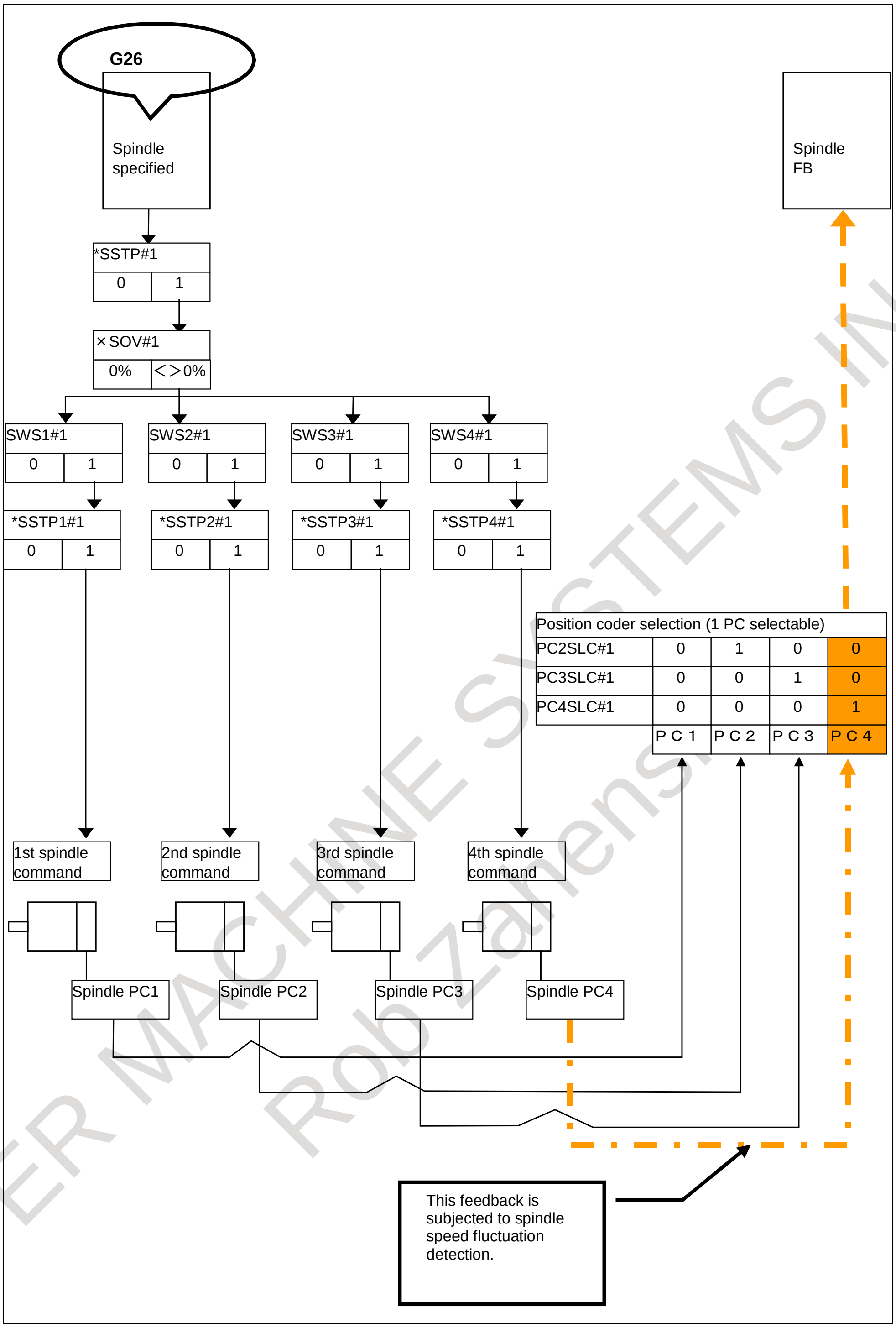
- Specified spindle : Indicates which path is specifying which spindle.
- Feedback pulse : Indicates the contents of the feedback pulse to be used, depending on a selected position coder.
- Feedback subjected to detection : Indicates the feedback subjected to the spindle speed fluctuation detection function.
- G26 : Indicates whether the spindle speed fluctuation detection function is enabled in which path.
- Detection : Indicates whether spindle speed fluctuation detection is possible in which path.

System configuration	Path	Specified spindle	Feedback pulse	Feedback subjected to detection	G26 (Multi-path)	Detection
Single path control + Non-multi-spindle control +1 spindle	—	Address S → 1st spindle	Feedback ← 1st spindle PC1	←○	○	○
Single path control + Multi-spindle control + 4 spindles	—	Address S → 1st spindle → 2nd spindle → 3rd spindle → 4th spindle	Feedback ← 1st spindle PC1 2nd spindle PC2 3rd spindle PC3 4th spindle PC4 (selected)	←× ←× ←× ←○	○	○
Multi-path control + Non-multi-spindle control +	Signal of spindle control for each path: Type A (*1.)					
	Path 1	Address S → 1st spindle	Feedback ← 1st spindle PC1	←○	○	○
	Path 2		Feedback ←	—	○	○
	Signal of spindle control for each path: Type A (*1.)					
	Path 1	Address S → 1st spindle	Feedback ← 1st spindle PC1	←○	×	×
	Path 2		Feedback ←	—	○	○
Multi-path control + Multi-spindle control + 2 spindles per path	Signal of spindle control for each path: Type C (*2.)					
	Path 1	Address S → 1st spindle → 2nd spindle	Feedback ← 1st spindle PC1 (selected) 2nd spindle PC2	←○ ←×	○	○
	Path 2	Address S → 1st spindle → 2nd spindle	Feedback ← 1st spindle PC1 2nd spindle PC2 (selected)	←× ←○	×	×
	Signal of spindle control for each path: Type C (*3.)					
	Path 1	Address S → 1st spindle → 2nd spindle	Feedback ← 1st spindle PC1 (selected) 2nd spindle PC2	←○ ←×	○	○
	Path 2	1st spindle 2nd spindle	Feedback ← 1st spindle PC1 2nd spindle PC2 (selected)	←× ←×	○	○

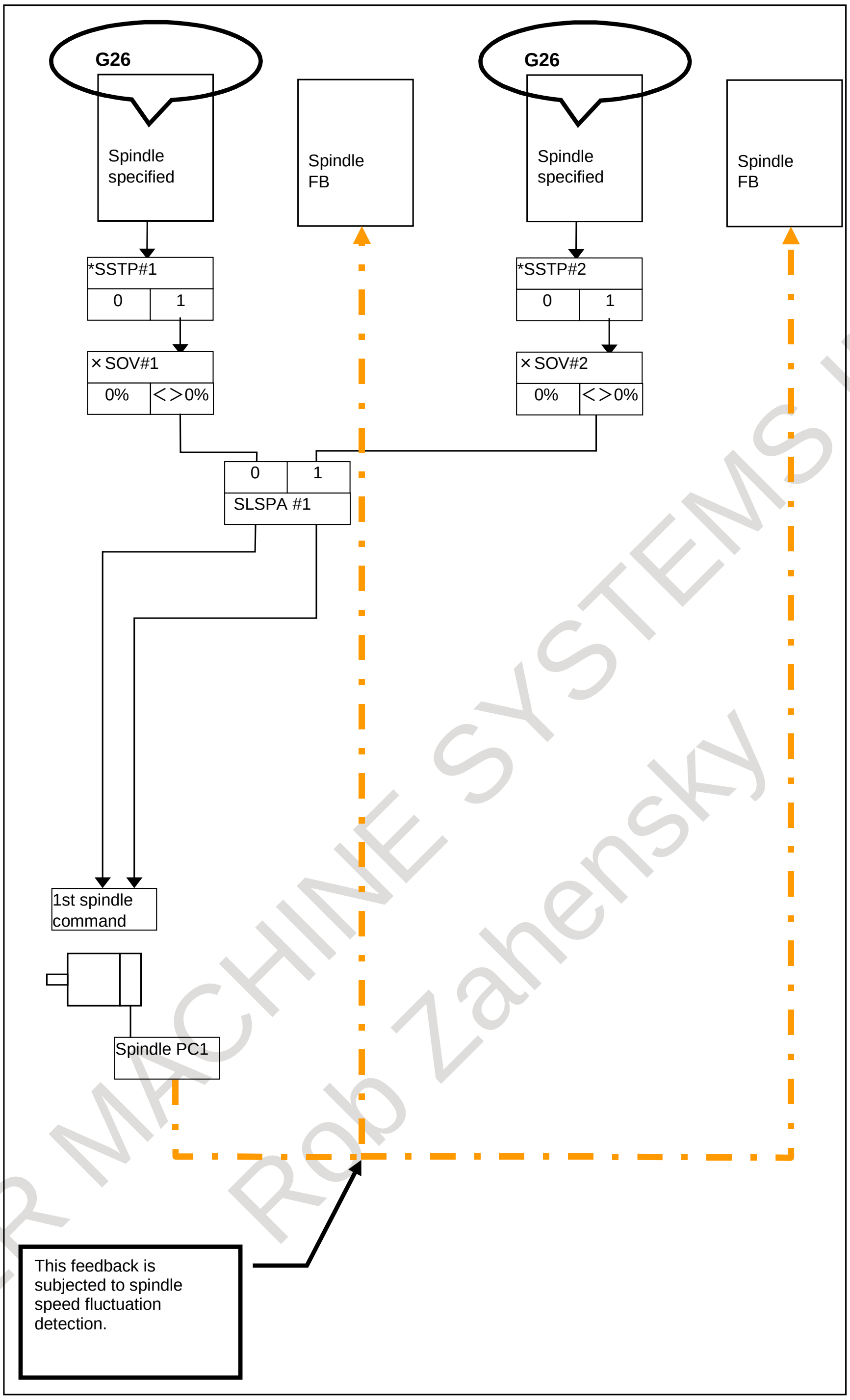
*1 The feedback pulse can be obtained from path 1 in this example.

*2 The signals for selecting spindle feedback of each path (SLPCA<Gn064.2>, SLPCB<Gn064.3>, SLPCC<Gn403.4>, SLPCD<Gn403.5>) are set to select feedback in each path, for both path 1 and path 2.

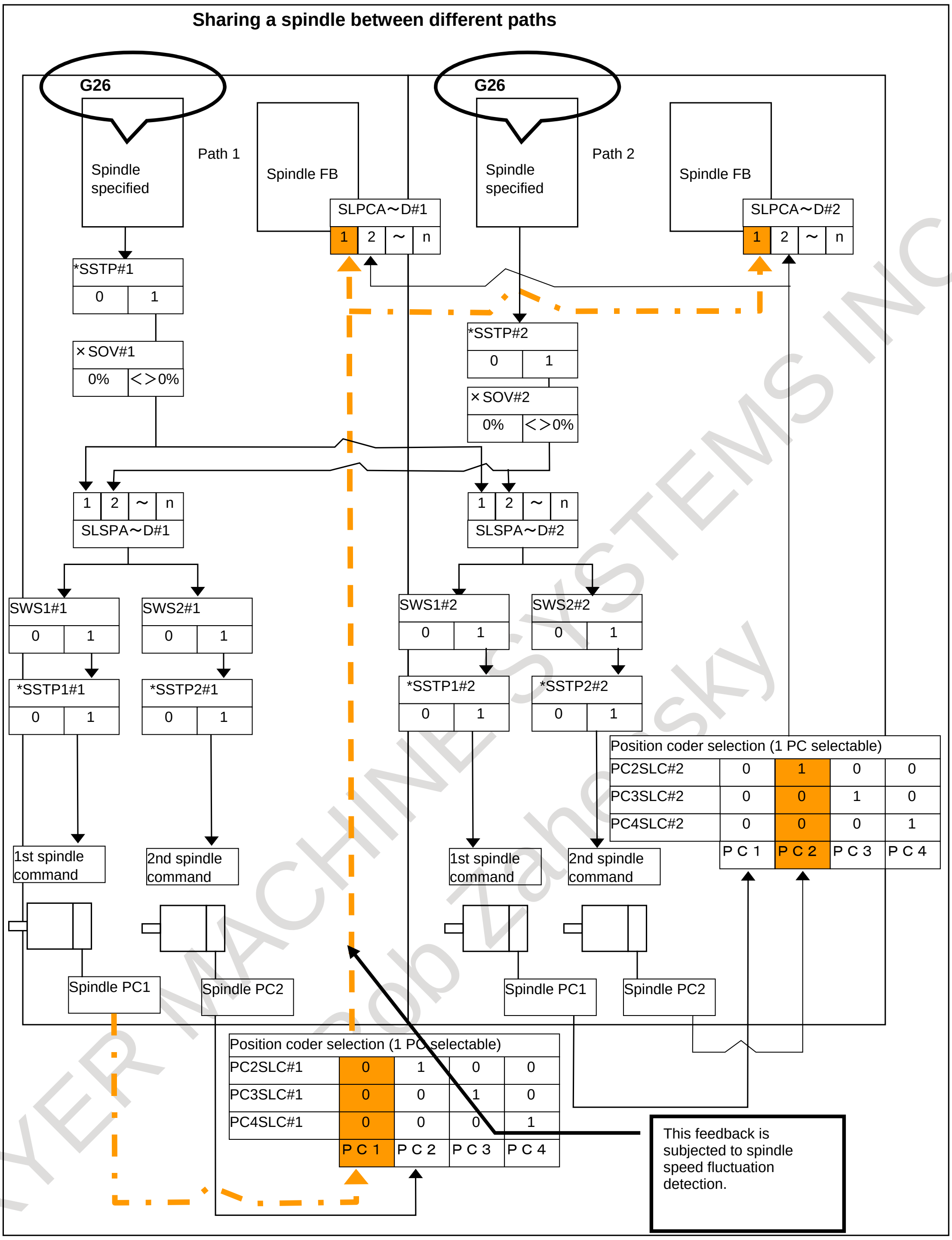
*3 The signals for selecting spindle feedback of each path (SLPCA<Gn064.2>, SLPCB<Gn064.3>, SLPCC<Gn403.4>, SLPCD<Gn403.5>) are set to select the feedback pulse of path 1 for path 2.



System controlling four spindles by one path



System controlling one spindle by two paths



Sharing a spindle between different paths

- About spindle speed fluctuation detection at power-on or cleared state

At power-on or cleared state, G25 mode (spindle speed fluctuation detection is disabled) is usually selected. If bit 0 (G26) of parameter No.4902 is set to 1, G26 mode (spindle speed fluctuation detection is enabled) is selected.

Placing the G25/G26 group in the cleared state requires the following parameter settings:

- Bit 6 (CLR) of parameter No. 3402 is set to 1
- Bit 0 (C08) of parameter No. 3407 is set to 1 (T series) / Bit 3 (C19) of parameter No. 3408 is set to 1 (M series)

About these parameters and G25/G26 mode state at power-on or reset, refer to following.

G26(No.4902#0)	CLR(No.3402#6)	(T series) C08(No.3407#0) (M series) C19(No.3408#3)	At power-on	At reset in G25 mode	At reset in G26 mode
0	0	0	G25	G25	G26
		1	G25	G25	G26
	1	0	G25	G25	G25
		1	G25	G25	G26
1	0	0	G26	G25	G26
		1	G26	G25	G26
	1	0	G26	G26	G26
		1	G26	G25	G26

- Parameters of spindle speed fluctuation detection at power-on

At power-on if bit 6 (UDPs) of parameter No.4900 is set to 1, parameters Nos. 4921 to 4924 (dp, dq, dr, di) are valid and parameters Nos. 4911 to 4914 (p, q, r, i) are invalid. Because of specified address P, Q, R, I in the machining program, value of parameters Nos. 4911 to 4914 (p, q, r, i) are changed and value of parameters Nos. 4921 to 4924 (dp, dq, dr, di) are not changed. So, parameters Nos. 4921 to 4924 (dp, dq, dr, di)are used as default parameter at power-on.

When parameters Nos. 4921 to 4924 (dp, dq, dr, di) are valid, if address P, Q, R, I are specified in the machining program even once as following,
G26 Pp Qq Rr Ii ;
the specified values are set for parameters Nos. 4911 to 4914 (p, q, r, i) and these parameters are valid as parameters of spindle speed fluctuation detection. If address P, Q, R, I are not specified in the machining program, parameters Nos. 4921 to 4924 (dp, dq, dr, di) are still valid. Examples of valid spindle speed fluctuation detection parameters are shown below.

(Example 1)

Current mode and valid parameters

G25 mode		p		q		r		i	
G26 mode	○	dp	○	dq	○	dr	○	di	○

↓ G26 Pp Qq Rr Ii ; ——— machining program

G25 mode		p	○	q	○	r	○	i	○
G26 mode	○	dp		dq		dr		di	

(Example 2)

G25 mode		p		q		r		i	
G26 mode	○	dp	○	dq	○	dr	○	di	○

↓ G26 Rr Ii ;

G25 mode		p		q		r	○	i	○
G26 mode	○	dp	○	dq	○	dr		di	

At cleared state, G25 / G26 mode changes according to parameter bit 0 (G26) of parameter No.4902, but valid parameters of spindle speed fluctuation detection don't change. And, When mode is change into

G25 mode, valid state of the parameters of spindle speed fluctuation detection is held. After that, when mode is change into G26 mode again, this valid state is applied. Examples are shown below.

(Example 3)

G25 mode		p		q		r		i	
G26 mode	○	dp	○	dq	○	dr	○	di	○

↓ G26 Rr Ii ;

G25 mode		p		q		r	○	i	○
G26 mode	○	dp	○	dq	○	dr		di	

↓ clear (bit 0 (G26) of parameter No.4902 is set to 1)

G25 mode		p		q		r	○	i	○
G26 mode	○	dp	○	dq	○	dr		di	

↓ G25 ;

G25 mode	○	p		q		r		i	
G26 mode		dp		dq		dr		di	

↓ G26 ;

G25 mode		p		q		r	○	i	○
G26 mode	○	dp	○	dq	○	dr		di	

(Example 4)

G25 mode		p		q		r	○	i	○
G26 mode	○	dp	○	dq	○	dr		di	

↓ G25 ;

G25 mode	○	p		q		r		i	
G26 mode		dp		dq		dr		di	

↓ clear (bit 0 (G26) of parameter No.4902 is set to 1)

G25 mode		p		q		r	○	i	○
G26 mode	○	dp	○	dq	○	dr		di	

NOTE

Alarm PS1305 is issued because value of specified address P, Q, R, I in the machining program is out of valid data range. At this time, about address of value out of valid data range, if type of parameters Nos. 4921 to 4924 (dp, dq, dr, di) are valid before machining program, type of parameters Nos. 4921 to 4924 (dp, dq, dr, di) are still valid.

Signal

Spindle speed fluctuation detection alarm signal SPAL <Fn035.0>

[Classification] Output signal

[Function] This signal indicates that the actual spindle speed is not within a tolerance to the specified speed.

[Output cond.] The signal becomes logical “1” when:

- The actual spindle speed goes out of tolerance to the specified speed.

The signal becomes logical “0” when:

- No alarm OH0704 condition has been detected for spindle speed fluctuation.
- An alarm condition is cleared by resetting the NC when the signal is logical 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn035								SPAL

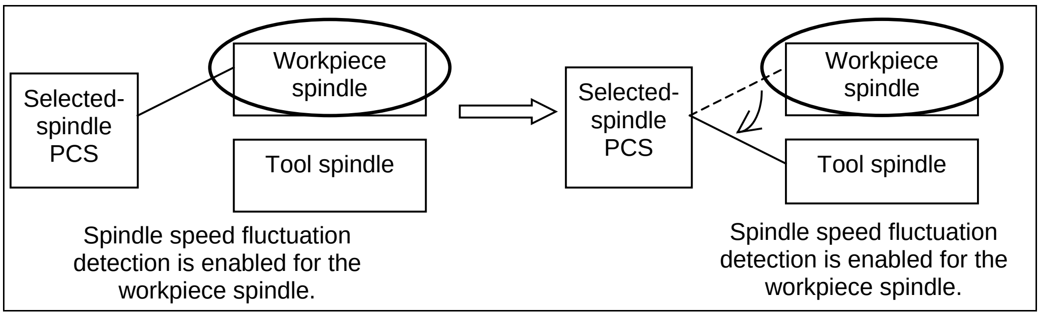
Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
4900	FDTs	UDPs		FDEs				FLRs

[Input type] Parameter input
[Data type] Bit spindle

#0 FLRs When the spindle speed fluctuation detection function is used, the unit of parameters No. 4911(q), No. 4912(r), No. 4921(dq), No. 4922(dr) and diagnoses No. 1591, No. 1592 (allowable ratio and fluctuation ratio) is:
0: 1%.
1: 0.1%.

#4 FDEs Spindle speed fluctuation detection function is:
0: Enabled.
1: Disabled.
If the position coder selection signal is selected for a spindle for which this parameter is 1, the target spindle of spindle speed fluctuation detection remains unchanged. Spindle speed fluctuation detection stays enabled for the spindle for which spindle speed fluctuation detection was enabled before the selection of the position coder selection signal.
An example is given below.
[Example]
Bit 4 (FDE) of parameter No.4900 (Workpiece spindle) = 0
Bit 4 (FDE) of parameter No.4900 (Tool spindle) = 1
Even when the position coder selection signal is switched from the workpiece spindle to the tool spindle, the target spindle of spindle speed fluctuation detection remains unchanged, that is, the function stays enabled for the workpiece spindle.



Note) The selected spindle varies depending on the state of the position coder selection signal.

NOTE

1. If bit 4 (FDE) of parameter No.4900 is 0 for all spindles, spindle speed fluctuation detection is enabled for the spindle selected with the position coder selection signal as is conventionally. If the bit 4 (FDE) of parameter No.4900 is 1 for all spindles, spindle speed fluctuation detection is enabled for the spindle selected with the position coder selection signal.
2. When the bit 4 (FDE) of parameter No.4900 is 0 for all spindles, setting the bit 4 (FDE) of parameter No.4900 for the spindle selected with the position coder selection signal to 1 does not cause an immediate change to the target spindle, which stays as the target until the next position coder selection signal is issued.
3. If the bit 4 (FDE) of parameter No.4900 for the first spindle is 1 and the first spindle is the tool spindle when the power is turned on, spindle speed fluctuation detection remains enabled for the first spindle until a spindle for which the bit 4 (FDE) of parameter No.4900 is 0 is selected.

#6 UDPs About parameters of spindle speed fluctuation detection at power-on:
0: Parameters Nos. 4911 to 4914 (p, q, r, i) are valid.
1: Parameters Nos. 4921 to 4924 (dp, dq, dr, di) are valid.

NOTE

This parameter (UDPs) is referenced at only power-on. So, even when this parameter (UDPs) is changed after power-on, valid parameters about spindle speed fluctuation detection are not changed.

#7 FDTs Spindle speed fluctuation detection is started:
0: When the actual spindle speed reaches a specified range or when the time specified with valid parameter (p(No.4914) or dp(No.4924)) elapses.
1: When the time specified with valid parameter (p(No.4914) or dp(No.4924)) elapses.

	#7	#6	#5	#4	#3	#2	#1	#0
4902								G26

[Input type] Parameter input
[Data type] Bit path

#0 G26 When power is turned on or when the control is cleared:
0: G25 mode (spindle speed fluctuation detection is disabled) is selected.
1: G26 mode (spindle speed fluctuation detection is enabled) is selected.

4911	Allowable speed ratio (q) used to assume that the spindle has reached a specified speed

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%, 0.1%
[Valid data range] 1 to 100, 1 to 1000

When the spindle speed fluctuation detection function is used, set an allowable speed ratio (q) used to assume that the spindle has reached a specified speed.

> **NOTE**
> The unit of data is determined by bit 0 (FLR) of parameter No. 4900.

4912	Spindle speed fluctuation ratio (r) for not issuing a spindle speed fluctuation detection alarm

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%, 0.1%
[Valid data range] 1 to 100, 1 to 1000

When the spindle speed fluctuation detection function is used, set a spindle speed fluctuation ratio (r) for not issuing an alarm.

> **NOTE**
> The unit of data is determined by bit 0 (FLR) of parameter No. 4900.

4913	Spindle speed fluctuation width (i) for not issuing a spindle speed fluctuation detection alarm

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999

When the spindle speed fluctuation detection function is used, set an allowable fluctuation width (i) for not issuing an alarm.

4914	Time (p) from the change of a specified speed until spindle speed fluctuation detection is started

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] msec
[Valid data range] 0 to 999999

When the spindle speed fluctuation detection function is used, set a time (p) from the change of a specified speed until spindle speed fluctuation detection is started. In other words, spindle speed fluctuation detection is not performed until a set time has elapsed after a specified speed is changed. However, when the actual spindle speed is assumed to have reached a specified value within a set time (p), spindle speed fluctuation detection is started.

4921	Allowable speed ratio (dq) used to assume that the spindle has reached a specified speed (at power-on)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%, 0.1%
[Valid data range] 1 to 100, 1 to 1000

When the spindle speed fluctuation detection function is used, set an allowable speed ratio (dq) used to assume that the spindle has reached a specified speed. To validate this parameter, it is necessary to set bit 6 (UDPs) of parameter No.4900 to 1. Then, at power-on, parameter No.4911 (q) is invalid, and parameter No.4921 (dq) is valid.

> **NOTE**
> 1 If address Q is specified in the G26 block after power-on, parameter No.4921 (dq) becomes invalid, and parameter No.4911 (q) becomes valid. If address Q is not specified in the G26 block after power-on, parameter No.4921 (dq) is still valid. Also, When alarm PS1305 is issued because specified address Q value is out of valid data range, parameter No.4921 (dq) is still valid.
> 2 The unit of data is determined by bit 0 (FLR) of parameter No.4900.

4922	Spindle speed fluctuation ratio (dr) for not issuing a spindle speed fluctuation detection alarm (at power-on)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%, 0.1%
[Valid data range] 1 to 100, 1 to 1000

When the spindle speed fluctuation detection function is used, set a spindle speed fluctuation ratio (dr) for not issuing an alarm. To validate this parameter, it is necessary to set bit 6 (UDPs) of parameter No.4900 to 1. Then, at power-on, parameter No.4912 (r) is invalid, and parameter No.4922 (dr) is valid.

> **NOTE**
> 1 If address R is specified in the G26 block after power-on, parameter No.4922 (dr) becomes invalid, and parameter No.4912 (r) becomes valid. If address R is not specified in the G26 block after power-on, parameter No.4922 (dr) is still valid. Also, When alarm PS1305 is issued because specified address R value is out of valid data range, parameter No.4922 (dr) is still valid.
> 2 The unit of data is determined by bit 0 (FLR) of parameter No.4900.

4923	Spindle speed fluctuation width (di) for not issuing a spindle speed fluctuation detection alarm (at power-on)

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999

When the spindle speed fluctuation detection function is used, set an allowable fluctuation width (di) for not issuing an alarm. To validate this parameter, it is necessary to set bit 6 (UDPs) of parameter No.4900 to 1. Then, at power-on or cleared state, parameter No.4913 (i) is invalid, and parameter No.4923 (di) is valid.

> **NOTE**
> If address I is specified in the G26 block after power-on, parameter No.4923 (di) becomes invalid, and parameter No.4913 (i) becomes valid. If address I is not specified in the G26 block after power-on, parameter No.4923 (di) is still valid. Also, When alarm PS1305 is issued because specified address I value is out of valid data range, parameter No.4923 (di) is still valid.

4924	Time (dp) from the change of a specified speed until spindle speed fluctuation detection is started (at power-on)

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] msec
[Valid data range] 0 to 999999

When the spindle speed fluctuation detection function is used, set a time (dp) from the change of a specified speed until spindle speed fluctuation detection is started. In other words, spindle speed fluctuation detection is not performed until a set time has elapsed after a specified speed is changed. However, when the actual spindle speed is assumed to have reached a specified value within a set time (dp), spindle speed fluctuation detection is started. To validate this parameter, it is necessary to set bit 6 (UDPs) of parameter No.4900 to 1. Then, at power-on, parameter No.4914 (p) is invalid, and parameter No.4924 (dp) is valid.

NOTE
If address P is specified in the G26 block after power-on, parameter No.4924 (dp) becomes invalid, and parameter No.4914 (p) becomes valid. If address P is not specified in the G26 block after power-on, parameter No.4924 (dp) is still valid. Also, When alarm PS1305 is issued because specified address P value is out of valid data range, parameter No.4924 (dp) is still valid.

Diagnosis data

1590	The index number of the spindle axis that is subjected to spindle speed fluctuation detection

[Data type] 2-word path

The index number of the spindle axis that is subjected to spindle speed fluctuation detection is indicated.

1591	Allowable speed ratio used to assume that the spindle has reached a specified speed

[Data type] Word path
[Unit of data] 1%, 0.1%
[Valid data range] 1 to 100, 1 to 1000

When the spindle speed fluctuation detection function is used, allowable speed ratio used to assume that the spindle has reached a specified speed is indicated.

NOTE
The unit of data is determined by bit 0 (FLR) of parameter No. 4900.

1592	Spindle speed fluctuation ratio for not issuing a spindle speed fluctuation detection alarm

[Data type] Word path
[Unit of data] 1%, 0.1%
[Valid data range] 1 to 100, 1 to 1000

When the spindle speed fluctuation detection function is used, spindle speed fluctuation ratio for not issuing an alarm is indicated.

> **NOTE**
> The unit of data is determined by bit 0 (FLR) of parameter No. 4900.

1593	Spindle speed fluctuation width for not issuing a spindle speed fluctuation detection alarm

[Data type] 2-word path
[Unit of data] min^{-1}
[Valid data range] 0 to 99999

When the spindle speed fluctuation detection function is used, allowable fluctuation width for not issuing an alarm is indicated.

1594	Time from the change of a specified speed until spindle speed fluctuation detection is started

[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 999999

When the spindle speed fluctuation detection function is used, time from the change of a specified speed until spindle speed fluctuation detection is started is indicated. In other words, spindle speed fluctuation detection is not performed until a set time has elapsed after a specified speed is changed. However, when the actual spindle speed is assumed to have reached a specified value within a set time, spindle speed fluctuation detection is started.

> **NOTE**
> Diagnosis data Nos.1590 to 1594 are "0" when spindle speed fluctuation detection function is disabled as follows.
> - G25 mode
> - There is no spindle selected with the spindle speed fluctuation detection
> - The specified spindle speed is 0 min^{-1}.
> - A search by program restart is in progress.
> - Spindle is under other than velocity control

Alarm and message

Number	Message	Description
OH0704	OVERHEAT:SPINDLE	Spindle is overheated by the spindle speed fluctuation detection function. <1> For heavy cutting, ease the cutting conditions. <2> Check whether the cutter has been blunted. <3> Check whether the spindle amplifier is malfunctioning.

Number	Message	Description
PS5305	ILLEGAL SPINDLE NUMBER	In a spindle select function by address P for a multiple-spindle control, 1) Address P is not specified. 2) Parameter No. 3781 is not specified to the spindle to be selected. 3) An illegal G code which cannot be commanded with an S_P_; command is specified. 4) The multi-spindle control cannot be used because the bit 1 (EMS) of parameter No. 3702 is 1. 5) The spindle amplifier number of each spindle is not set in parameter No. 3717. 6) A prohibited command for a spindle was issued (parameter No. 11090). 7) An invalid value is set in parameter No. 11090.

Note

NOTE

1 Even if the conditions to start the spindle speed fluctuation detection function are satisfied, the spindle speed fluctuation detection function is not performed under the following conditions:
- The specified spindle speed is 0 min^{-1}.
- A search by program restart is in progress.

2 If the spindle stop signal *SSTP<Gn029.6> is set to “0”, detection is not performed.

3 If any of the spindle stop signals *SSTP1 to *SSTP4<Gn027.3, Gn027.4, Gn027.5, Gn026.6> is set to “0”, detection is not performed on the corresponding spindle.

4 If alarm OH0704 "OVERHEAT:SPINDLE" is raised, a single-block stop occurs in automatic operation. The spindle overheat alarm is displayed on the screen. In addition, the spindle speed fluctuation detection alarm signal SPAL is output (the signal is set to “1” in the alarm state).
This signal is cleared to “0” by a reset.

5 If the cause of alarm OH0704 is not eliminated by a reset, the alarm will be raised again.

6 Alarm OH0704 can be suppressed across an allowable variation width specified in parameter No.4913. If the actual spindle speed continues to be 0 min^{-1} for one second or longer, however, an alarm will be raised.

7 The raising of alarm OH0704 does not automatically cause the spindle to stop.

8 G25 and G26 must be specified in a predetermined format. Avoid specifying the preparatory function, auxiliary functions, and the like together with G25 or G26.

9 A serial spindle signal that can affect the actual spindle speed should be manipulated in the G25 mode.

10 If a spindle is under velocity control, spindle speed fluctuation detection is carried out.

11 Avoid making a spindle orientation operation while the spindle speed fluctuation detection function is enabled.

12 If address P is used to select a spindle while the multi-spindle control is carried out, avoid specifying S_P_; together with G26. If this is attempted, alarm PS5305 will be raised.

NOTE

13 For a serial spindle, carefully check the gear count selection statuses of clutch/gear signals (CTH1, CTH2) and gear selection signals (GR3O, GR2O, GR1O (type M)/GR1, GR2(type T). The spindle speed fluctuation detection function does not normally work if the gear count statuses of the two types of signals are different.

CNC system							Serial spindle side	
Lathe system、Machining center system			Machining center system					
GR2	GR1	Selected gear	GR3O	GR2O	GR1O	Selected gear	CTH1	CTH2
0	0	1st stage	0	0	1	1st stage	0	0
0	1	2nd stage	0	1	0	2nd stage	0	1
1	0	3rd stage	1	0	0	3rd stage	1	0
1	1	4th stage					1	1

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	Path spindle control
	Spindle speed control
	Spindle output control by the PMC
	Multi-spindle control

FANUC AC SPINDLE MOTOR α*i* series PARAMETER MANUAL (B-65280EN)

10.20 SPINDLE CONTROL WITH SERVO MOTOR

Overview

Servo motors can be controlled by spindle functions like spindle rotation commands or rigid tapping.

(1) Spindle control with servo motor
Rotation commands (S command) can be used to control the speed of spindles by regarding servo motors as spindles (live tools). No reference position return is necessary to switch between rotation and positioning commands.

(2) Spindle indexing
There are two types of spindle indexing. The first type allows the next-block command to be executed before spindle indexing is finished. The second type allows the next block to be executed only after spindle indexing is completed.
With the first type, it is possible to issue commands to axes other than the axis for which a spindle indexing command has been issued, before the next command is issued to the axis. The next command can be issued only after it is confirmed that spindle indexing is completed. Using this function can reduce the wait time. In addition, an axis can be stopped at a specified point by issuing a spindle indexing command to the axis when the spindle is rotating.

(3) Rigid tapping with servo motor
Rigid tapping can be carried out by regarding a servo motor as a rotation axis.

(4) Threading, feed per revolution, and constant surface speed control
Threading, feed per revolution, and constant surface speed control can be carried out by regarding a servo motor as a spindle.

(5) Spindle output control with PMC
Spindle output control with PMC can be carried out by regarding a servo motor as a spindle controlled axis.

Notes

(1) Enable the spindle serial output (bit 5 (SSN) of parameter No,8133 is 0) and multi-spindle control (bit 3 (MSP) of parameter No,8133 is 1).
If this function is used under the conditions below, the multi-spindle control option is not enabled.
- Two or more spindles are not used on any path.

- The spindle gear type is T.
- No G code (G96.1, G96.2, G96.3, or G96.4) commands are used.
- Spindle indexing is not used.

(2) This function handles a servo motor used as a spindle controlled axis as one of controlled spindles and one of controlled axes.
(3) When carrying out rigid tapping with servo motor, enable the rigid tapping (bit 3 (NRG) of parameter No,8135 is 0) and the canned cycle (bit 4 (NCD) of parameter No,8137 is 0).
(4) When carrying out constant surface speed control, enable the constant surface speed control (bit 0 (SSC) of parameter No,8133 is 1).

Spindle motors and supported functions

Spindle / Function	Conventional spindle control	Spindle control with servo motor
Threading/feed per revolution	○	○
Polygon machining (T series)	○	×(*1)
Spindle speed fluctuation detection	○	×
Spindle synchronous control	○	×
Spindle simple synchronous control	○	×
Polygon machining with two spindles	○	×
Spindle orientation Multi-point orientation Spindle output switching	○	×
Inter-path spindle control	○	○
Constant surface speed control	○	○
Multi-spindle control	○	○
Rigid tapping	○	○
Spindle output control with PMC	○	○
Actual spindle speed output (T series)	○	○
Spindle indexing	×	○

NOTE
Servo motor can be used as tool rotation axis. (*1)

10.20.1 Spindle Control with Servo Motor

Specification

- Command with a program

This function supports two modes; SV speed control mode (a spindle rotation command (S command) is enabled for the servo motor-controlled rotation axis) and position control mode (ordinary positioning is enabled). Controlling servo motor rotation requires issuing an SV speed control mode ON command (G96.4). Once the command is issued, the S command for the servo motor remains enabled until the mode is canceled. No positioning command is usable if the SV speed control mode is ON.
Performing positioning requires canceling the SV speed control mode. Canceling the SV speed control mode (setting the position control mode to ON) requires issuing a spindle indexing command (G96.1 or G96.2). In the position control mode, the S command is disabled as in the ordinary servo axis state. However, information about the S command is preserved. So, when the SV speed control mode becomes ON, the motor starts rotating at the speed already specified.

- Command with a signal

The SV speed control mode signal <Gn521> can also be used to specify whether to turn ON or OFF the SV speed control mode.
If the SV speed control mode is turned ON using the signal and then OFF using a program command, turning it ON again requires re-entering the signal or re-issuing the G96.4 command.

The SV speed control mode in-progress signal <Fn521> can be used to check whether the SV speed control mode is in effect.
Turning the SV speed control mode signal OFF during rotation performs spindle indexing and then turns the SV speed control mode OFF. Spindle indexing is executed with R0 (absolute position 0).
Specify an M code not involved in buffering as an M code to be used as a command for switching the SV speed control mode (parameters Nos. 3411 to 3420 and 11290 to 11299).

Format

G96.4 P_ ;	SV speed control mode ON
M03 (M04) S_ P_ ;	Rotation command

S: Spindle speed [min^{-1}] (numeric value of up to five digits)
P: Spindle selection with multi-spindle control

Using G96.4 and a spindle selection command P can turn the SV speed control mode ON for each axis separately. Likewise, using a spindle indexing command (G96.1/G96.2) and a spindle selection command P can cancel the SV speed control mode, that is, turn on the position control mode ON, for each axis separately.
To specify “G96.4 P_ ;”, use an independent block.
However, it is possible to specify "G96.4 P_ S_ ;".

Explanation

- Command

(1) Spindle speed command output
Set up the spindle speed command in the same way as for the ordinary speed command (S command). However, turn the SV speed control mode ON before issuing the rotation command (S command). When performing positioning, cancel the SV speed control mode, and select the position control mode.
The sequence by the following input signals is unnecessary.
*ESPA, MRDYA, and SFRA

(2) Condition for stopping spindle speed output
Commands once output to a spindle become 0 if *SSTP becomes 0 or if a command (such as S0) that makes the spindle speed command output 0 is issued. Also, issuing a spindle indexing command (G96.1/G96.2) makes the spindle speed command output 0. In addition, an emergency stop condition and servo alarm bring the spindle to a stop.
With M05, the CNC does not make the command output to the spindle 0.

(3) Stopping rotation by spindle indexing
Issuing a command that specifies a position enables the rotating axis to stop at the specified position. For details, see Subsection of "Spindle Indexing Function."

(4) Maximum speed
The maximum speed that can be specified is usually 2777 min^{-1}.
However, setting bit 3 (IRC) of parameter No. 1408 to 1 may be able to increase the limit to about 27770 min^{-1} depending on the performance of the related motor and detector. (when IS-B is used.)

Depending on the increment system, the maximum speed is as below.

Increment system	Bit 7 (IESP) of parameter No. 1013 = 0	Bit 7 (IESP) of parameter No. 1013 = 1
IS-B	27750min^{-1}	27750min^{-1}
IS-C	2777min^{-1}	27750min^{-1}

It is assumed that bit 3 (IRC) of parameter No. 1408 is set to 1 in all cases.

Example command for switching between rotation control and position control

Spindle name	Spindle select P code (parameter No. 3781)	Servo motor axis address
S1	P1	C

Command with a program (SV speed control mode in-progress signal (Fn521))

Program command	SV speed control mode ON/OFF	Operation
G96.4 P1 ;	ON (SV speed control mode in-progress signal (C) =1)	SV speed control mode ON (C).
M03 S100 P1 ;	ON	The servo motor rotation axis C rotates in the normal direction at 100 [min^{-1}].
:	:	:
G96.1 P1 R0 ;	OFF (SV speed control mode in-progress signal (C) = 0)	The servo motor rotation axis C stops at C = 0 (spindle indexing).
G00 C180.0 ;	OFF (SV speed control mode in-progress signal (C) = 0)	Positioning at C = 180 degrees.
M03 S200 P1 ;	OFF (SV speed control mode in-progress signal (C) = 0)	M03 S200 is stored without causing the servo motor to rotate.
G96.4 P1 ;	ON (SV speed control mode in-progress signal (C) = 1)	The servo motor rotation axis C rotates in the normal direction at 200[min^{-1}]. SV speed control mode ON (C).

Signal-controlled mode switching

Program command	SV speed control mode ON/OFF	Operation
M15 ;	ON (SV speed control mode in-progress signal (C) = 1)	SV speed control mode is turned ON (C) with M code.
M03 S100 P1 ;	ON (SV speed control mode in-progress signal (C) =1)	Servo motor rotation axis C rotates in normal direction at 100 [min^{-1}].
:	:	:
G96.1 P1 R0 ;	OFF (SV speed control mode in-progress signal (C)=0)	Servo motor rotation axis C stops at C = 0 (spindle indexing).
G00 C180. ;	OFF (SV speed control mode in-progress signal (C)=0)	Positioning at C = 180 degrees.
M15 ;	ON (SV speed control mode in-progress signal (C)=1)	SV speed control mode is turned ON (C) with M code.
M03 S100 P1 ;	ON	Servo motor rotation axis C rotates in normal direction at 100 [min^{-1}].

- Multi-spindle control and commands from another path

Controlling the rotation of a spindle in a path that has another spindle (such as a servo motor used as a live tool) requires the multi-spindle control. The path spindle control function can handle commands from another path. The address P-based spindle select function for multi-spindle control can be used to select spindles in another path.

Example of program commands are as follows.

Example 1: (Spindle selection with address P)

Bit 3 (MPP) of parameter No. 3703 = 1: A spindle is selected with address P.

Spindle configuration (S1 = first spindle and S2 = second spindle)

Path 1	Path 2
S1 (spindle axis)	S3 (spindle axis)
S2 (servo axis)	S4 (servo axis)

Setting address P for spindle selection in multi-spindle control

Parameter	Path 1	Path 2
3781	11 (S1)	21 (S3)
	12 (S2)	22 (S4)

Program example

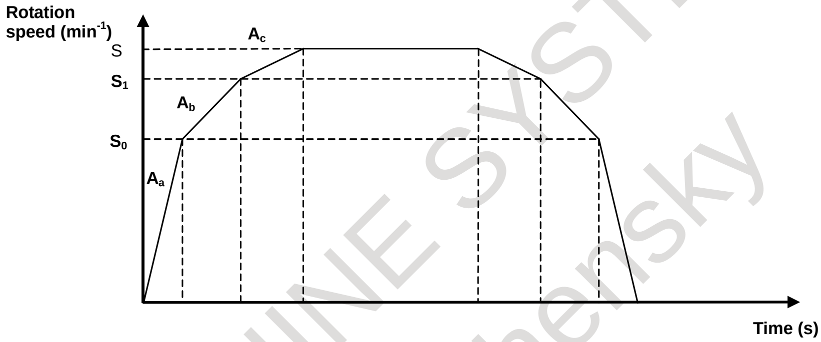
Command issuing path	Program	Operation
1	M03 S1000 P12 ;	S2 rotates in normal direction at 1000 min^{-1}.
1	M03 S1500 P22 ;	S4 rotates in normal direction at 1500 min^{-1}.
2	M04 S1500 P11 ;	S1 rotates in reverse direction at 1500 min^{-1}.

- Operation (automatic/manual) of spindle control with servo motor axis

Once a spindle control with servo motor axis has returned to its reference position, it can operate in the same manner as in the ordinary servo axis or spindle rotation control mode. To operate the a spindle control with servo motor axis manually, keep the SV speed control mode OFF.

- Acceleration/deceleration (time constant)

It is possible to change the spindle acceleration/deceleration specified in a rotation command according to the speed of the spindle.
Acceleration/deceleration can be switched at two points of speed, using parameters S0 and S1 (switching speed). In addition, parameters Aa, Ab, and Ac are available to set up three acceleration/deceleration spans.



S_0	Setting of parameter No. 11020 (acceleration/deceleration is switched at rotation speed S0 (min^{-1})).
S_1	Setting of parameter No. 11021 (acceleration/deceleration is switched at rotation speed S1 (min^{-1})).
S	Specified rotation speed (min^{-1}).
Aa	Setting of parameter No. 11030 (acceleration/deceleration (min^{-1}/s) used between rotation speeds 0 and S0).
Ab	Setting of parameter No. 11031 (acceleration/deceleration (min^{-1}/s) used between rotation speeds S0 and S1).
Ac	Setting of parameter No. 11032 (acceleration/deceleration (min^{-1}/s) used between rotation speeds S1 and motor's maximum).

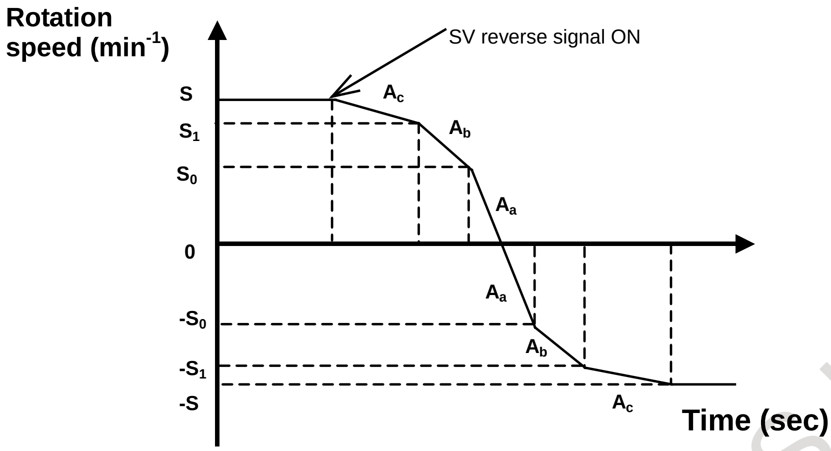
Determine the setting of each parameter according to the torque characteristic of the motor.

- Acceleration/deceleration after interpolation

Acceleration/deceleration after interpolation is available in the SV speed control mode. Bit 1 (TCR) of parameter No. 11001 can be used to select a time constant type, that is, parameter No. 1622
(Time constant of acceleration/deceleration in cutting feed for each axis) or parameter No. 11016 (Time constant of acceleration/ deceleration in SV speed control mode for each axis).

- Direction of rotation

It is possible to change the polarity of the spindle speed output voltage, using bits 6 (CWM) and 7 (TCM) of parameter No. 3706.
It is also possible to reverse the direction of spindle rotation, using the SV reverse signal <Gn523>.
These functions are usable in the rotation mode and during rigid tapping. Changing the signal to 1/0 during rotation causes the spindle to decelerate, reverse, and then accelerate.



- Reference position return

Before issuing the first command, be sure to make a reference position return.

- Display

Bit 3 (NDF) of parameter No. 3115 can be used to specify whether to display the actual speed. In addition, bits 0 (NDP) and 1 (NDA) of parameter No. 3115 can be used to specify whether to display, respectively, the current position and the remaining amount of movement.

- Setting of a rotation axis with a servo motor

(1) Axis setting
For the servo axis number of the axis to be set as a rotation axis with a servo motor (parameter No. 1023), set a value as a servo motor as usual. Specify which servo motor to use as a spindle controlled axis, using bit 7 (SRV) of parameter No. 11000. Specify which spindle to use as the spindle controlled axis with the servo motor, by setting the spindle number in parameter No. 11010. For the selected spindle, reset the spindle amplifier number (parameter No. 3717) to 0.
In multi-spindle control, set bit 3 (MPP) of parameter No. 3703 to 1 to specify that program commands be used to select a spindle.
Also set parameter No. 3781 with the P code for selecting a spindle.

(2) Using a command to specify a spindle speed requires adjusting the following parameters.

Parameter No. 11013:	Positioning deviation limit in movement
Parameter No. 11014:	Positioning deviation limit in the stop state
Parameter No. 11015:	Maximum motor speed

(3) Rotation axis setting
Using bits 0 (ROTx) and 1 (ROSx) of parameter No. 1006, select rotation axis type A.
Using the rollover function, round the absolute coordinate values to be displayed to an amount of movement per rotation specified in parameter No. 1260 in order to prevent rotation axis coordinate overflow. Also round the relative coordinate values to be displayed to an amount of movement per rotation by setting bit 2 (RRLx) of parameter No. 1008 to 1.
The rollover function is enabled by setting bit 0 (ROAx) of parameter No. 1008 to 1 (for the rotation axis).

(4) Extension of maximum speed
When using spindle control with servo motor, set 1 to bit 3 (FEX) of parameter No.8003.

- Example of parameter setting

The following examples show typical parameter settings. These typical parameter settings should be used unless otherwise stated.

(1) Spindle controlled axis setting
Let the fourth axis of the CNC be the C axis. Assuming that the fourth axis is connected to the second spindle, let us use the second spindle as a spindle controlled axis (servo motor).
No.11000#7 (C) = 1

No.11010 (C) = 2 (a spindle number is specified for a controlled axis assigned to the spindle controlled axis.)

(Set up No. 3717 (S2) = 0.)

Given below are the least command increment, detection unit, and the amount of movement per rotation for the C axis.

Least command increment = $\frac{L \times CMR}{Q \times DMR}$

Detection unit = $\frac{\text{Least command increment}}{CMR} = \frac{L}{Q \times DMR}$

Amount of movement per tool axis rotation =

$\frac{360}{\text{Least command increment}}$

where

L: Amount of tool movement per motor rotation (360 × speed increment ratio) [deg]
For example, if the servo motor is connected directly to the spindle (live tool), L = 360. For example, if the speed must be increased by twice, L = 720.

Q: Number of pulses per pulse coder rotation
(For the serial pulse coder, Q = 1000000.)
The least command increment mentioned above is specific to the C axis and is determined independently of bits 0 (IS-A) and 1 (IS-C) of parameter No 1013.
Reset both bits 0 (IS-A) and (IS-C) of parameter No 1013 to 0 to select IS-B.
If the servo motor is connected directly to the spindle (live tool):

Least command increment= $\frac{360 \times 1}{1000000 \times 36/100} = 0.001$ [deg]

Detection unit = 0.001 [deg]

Amount of movement per tool axis rotation =360.0 [deg]

So, the required parameter setting must be:

Parameter No. 11011 (C) = 360.000 (amount of movement per spindle controlled axis rotation)

In addition, the acceleration/deceleration values for rotation control must be specified in parameters Nos. 11020 to 11032.

Let us assume that the axis parameters be specified for the fourth axis, as follows:

(2) Servo parameter setting

Let us assume: CMR = 1 and DMR = 36/100

(The capacity of the reference counter will be 360000.)

No.1820 (C) = 2	(CMR)
No.1821 (C) = 360000	(Reference counter capacity)
No.2084 (C) = 36	(DMR numerator)
No.2085 (C) = 100	(DMR denominator)

If the tool axis-to-motor gear ratio is 1:1:

No.11015(C) = 5000	(Maximum motor speed)
No.3741(S2) = 5000	(Maximum speed for gear 1)

For the other servo parameters, use the respective typical settings.

(3) Full closed system setting

When using spindle control in a full closed system, set bit 1 (FSR) of parameter No. 11000 to 1.

Signal

SV speed control mode signals SRVON1 to SRVON8<Gn521>

[Classification] Input signal

[Function] These signal switch the respective axes to the SV speed control mode.

[Operation] Setting these signals to 1 cause the respective axes to enter the SV speed control mode.
Resetting these signals to 0 cause the respective axes to exit the SV speed control mode.
Resetting the signals to 0 during axis rotation causes the respective axes to start spindle indexing, to stop at the origin (R0), and to exit the SV rotation mode.

> **NOTE**
> 1 Even if these signals are 1, the spindle control function remains disabled when a program command turns the SV speed control mode OFF (the SV speed control mode in-progress signal Fn521 becomes 0). To enable it, re-set these signals to 1.
> 2 Specify an M code not involved in buffering as an M code to be used as a command for switching the SV speed control mode (parameters Nos. 3411 to 3420 and 11290 to 11299).

SV reverse signals SVRVS1 to SVRVS8<Gn523>

[Classification] Input signal

[Function] These signals cause the respective axes to reverse their rotation direction in the SV speed control mode.

[Operation] Setting these signals to 1 cause the respective axes to reverse their rotation direction.
Resetting these signals to 0 cause the respective axes to resume their normal rotation direction.

> **NOTE**
> These signals are effective even in rigid tapping.

SV speed control mode in-progress signals SVREV1 to SVREV8<Fn521>

[Classification] Output signal

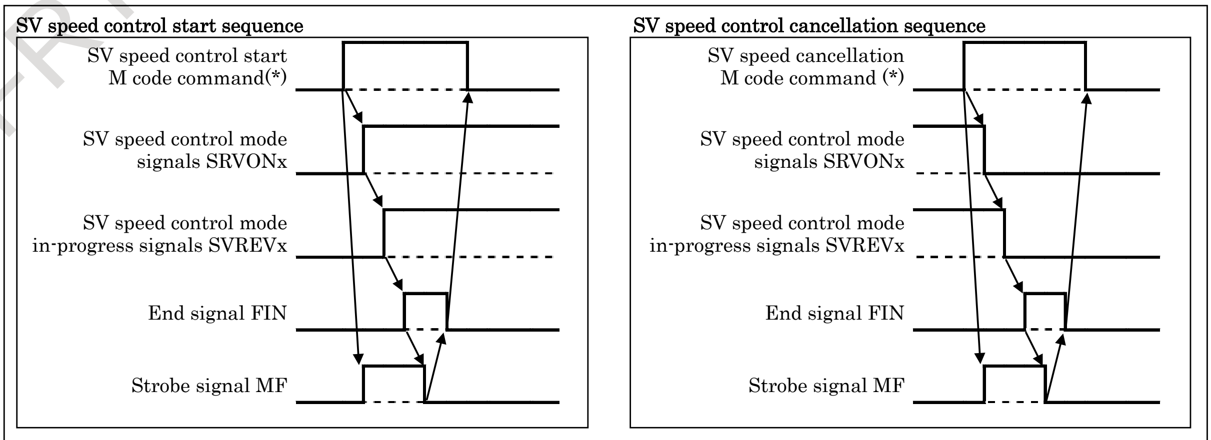
[Function] These signals inform that the respective axes are in the SV speed control mode.

[Output cond.] These signals are 1 if:

- The respective axes are in the SV speed control mode.

These signals are 0 if:

- The respective axes are in the SV speed control mode, or
- The respective axes are in the position control mode.



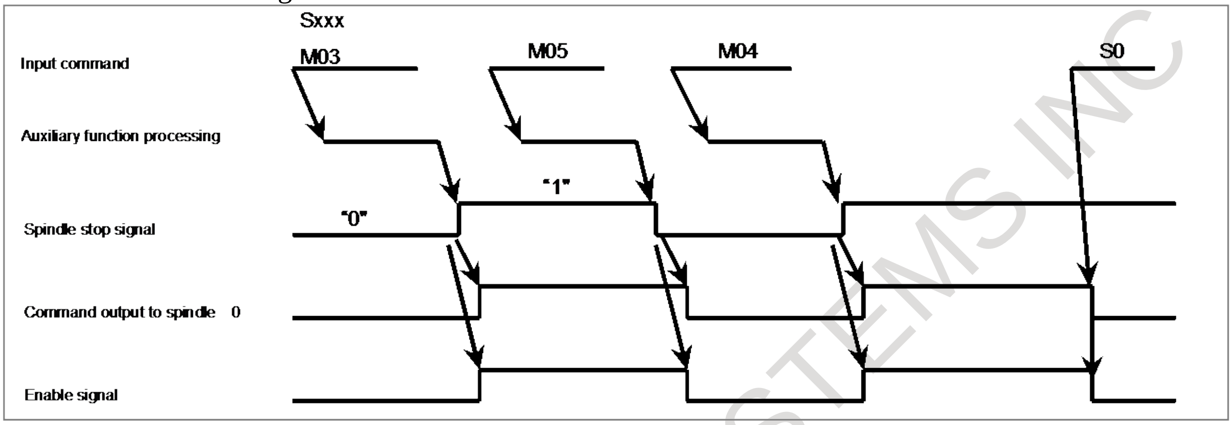
(*) Specify an M code not involved in buffering as an M code to be used as a command for switching the SV speed control mode.

Spindle stop signal *SSTP<Gn029.6>

[Classification] Input signal

[Function] This signal stops the command output to the spindle.

[Operation] When this signal becomes 0, the command output to the spindle becomes 0, resulting in the enable signal becoming 0. So, M05 is not output. When this signal becomes 1, the command output to the spindle resumes its previous value, resulting in the enable signal becoming 1.



The timing chart above is illustrative only. To be more precise, follow a timing chart that meets the specification of the speed control unit for the spindle motor in use.

- Keep this signal set to 1 unless it is used.
- The CNC uses the commands M03, M04, and M05 to send only code and strobe signals.

Individual spindle stop signals
*SSTP1<Gn027.3> *SSTP2<Gn027.4> *SSTP3<Gn027.5> *SSTP4<Gn026.6>

[Classification] Input signal

[Function] These signals are usable only during multi-spindle control.
They cause the respective spindles to stop.

*SSTP1 1: Does not make the output to the first spindle 0 min^{-1}.
0: Makes the output to the first spindle 0 min^{-1}.

*SSTP2 1: Does not make the output to the second spindle 0 min^{-1}.
0: Makes the output to the second spindle 0 min^{-1}.

*SSTP3 1: Does not make the output to the third spindle 0 min^{-1}.
0: Makes the output to the third spindle 0 min^{-1}.

*SSTP4 1: Does not make the output to the fourth spindle 0 min^{-1}.
0: Makes the output to the fourth spindle 0 min^{-1}.

Spindle speed override signals SOV0 to SOV7<Gn030>

[Classification] Input signal

[Function] These signals enable an override of 0% to 254% of the S command value sent to the CNC to be applied to the spindle in 1% increments.

[Operation] Specify an override value in binary, using the 8 bits SOV0 to SOV7.
The spindle speed override specified with these signals becomes disabled, and an override of 100% is applied to the spindle speed during:

- Tapping cycle (for the M series, G84 and G74 or, for the T series, G84 and G88)
- Threading mode (for the M series, G33 or, for the T series, G32, G92, and G76)
  → Specify an override of 100% unless this function is used.
  (An override of 0% is applied, preventing spindle rotation.)

Spindle enable signals ENB<Fn001.4> ENB2<Fn038.2> ENB3<Fn038.3> ENB4<Fn039.1>

[Classification] Output signal

[Function] These signals inform whether there is an output to the second, third, or fourth spindle if the multi-spindle control is available.
The signals can be used as a condition for stopping the analog spindle and in PMC ladder sequences related to rigid tapping.
(See Subsection, "Rigid Tapping.")

[Output cond.]

ENB 1 if the output to the first spindle control unit is not 0.
0 if the output to the first spindle control unit is 0.

ENB2 1 if the output to the second spindle control unit is not 0.
0 if the output to the second spindle control unit is 0.

ENB3 1 if the output to the third spindle control unit is not 0.
0 if the output to the third spindle control unit is 0.

ENB4 1 if the output to the fourth spindle control unit is not 0.
0 if the output to the fourth spindle control unit is 0.

Multi-spindle address P signals MSP00 to MSP15<Fn160.0 to Fn161.7>

[Classification] Output signal

[Function] These signals output the P value most recently specified by the S_ P_ ; command.

[Output cond.] If the address P-based multi-spindle control is enabled (bit 3 (MPP) of parameter No. 3703 =1), the P value specified by the S_ P_ ; command is output. If no S_ P_ ; has been issued since application of the power, the initial P value specified in parameter No. 3775 is output.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn026		*SSTP4						
Gn027			*SSTP3	*SSTP2	*SSTP1			
Gn029		*SSTP						
Gn030	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0
Gn521	SRVON8	SRVON7	SRVON6	SRVON5	SRVON4	SRVON3	SRVON2	SRVON1
Gn523	SVRVS8	SVRVS7	SVRVS6	SVRVS5	SVRVS4	SVRVS3	SVRVS2	SVRVS1
Fn001				ENB				
Fn038					ENB3	ENB2		
Fn039							ENB4	
Fn160	MSP07	MSP06	MSP05	MSP04	MSP03	MSP02	MSP01	MSP00
Fn161	MSP15	MSP14	MSP13	MSP12	MSP11	MSP10	MSP09	MSP08
Fn521	SVREV8	SVREV7	SVREV6	SVREV5	SVREV4	SVREV3	SVREV2	SVREV1

Notes

- Unlike spindle motors, servo motors stop if a servo alarm, emergency stop, or machine lock condition occurs.
- Specify an M code not involved in buffering as an M code to be used as a command for switching the SV speed control mode (parameters Nos. 3411 to 3420 and 11290 to 11299).
- M code for switching the SV speed control mode must be specified in an independent block.
  When using multiple M commands in a single block, the M code must be specified as the first M command.
- G96.1, G96.2, G96.3, G96.4, and M code for switching the SV speed control mode must be specified in the path to which the spindle of interest belongs.
- The actual maximum motor speed is determined by applying a feed gear factor to the maximum motor speed specified in parameter No. 11015.
- If bit 3 (IRC) of parameter No. 1408 is 0, the malfunction prevent function may work at a rotation speed of about 2778 min-1. To allow the motor to run at 2778min-1 or faster, set bit 3 (IRC) of parameter No. 1408 to 1. If the bit is set to 1, the command can specify a rotation speed of up to about 27778 min^{-1}.
- This function must be set for type T gears.
- G96.1, G96.2, G96.3, G96.4 cannot be commanded for settings other than multi-spindle control type P.
- Feed axis synchronous control or synchronous control cannot be performed on an axis that is in SV rotation control mode.
- As for the axis used for spindle control with a servo motor, disable the setting of bit 7 (ALGx) of parameter No. 1814, which adjusts the loop gain in Cs contour control mode to the loop gain of the Cs contour control axis, and the setting of parameter No. 3900 and after, which sets the loop gain of the axis to interpolate with the Cs contour control axis. Otherwise, the loop gain of the servo axis will change suddenly if the system enters Cs contour control mode during rotation control.
- When using spindle control with servo motor, set 1 to bit 3 (FEX) of parameter No.8003.

Parameter

The major related parameters are described below.

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx Setting linear or rotation axis.
#1 ROSx

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)

ROSx	ROTx	Meaning
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No. 1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

1022	Setting of each axis in the basic coordinate system

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

To determine a plane for circular interpolation, cutter compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane) specify which of the basic three axes (X, Y, and Z) is used for each control axis, or a parallel axis of which basic axis is used for each control axis.
A basic axis (X, Y, or Z) can be specified only for one control axis.
Two or more control axes can be set as parallel axes for the same basic axis.

Setting	Meaning
0	Rotary axis (Neither the basic three axes nor a parallel axis)
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

In general, the increment system and diameter/radius specification of an axis set as a parallel axis are to be set in the same way as for the basic three axes.

1023	Number of the servo axis for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 80

This parameter associates each control axis with a specific servo axis. Specify values 1+8n, 2+8n, 3+8n, 4+8n, 5+8n, and 6+8n (n = 0, 1, 2, …, 9) like 1, 2, 3, 4, 5, …, 77, and 78.

The control axis number is the order number that is used for setting the axis-type parameters or axis-type machine signals

- With an axis for which Cs contour control/spindle positioning is to be performed, set -(spindle number) as the servo axis number.
  Example)
  When exercising Cs contour control on the fourth controlled axis by using the first spindle, set -1.
- For tandem controlled axes or electronic gear box (EGB) controlled axes, two axes need to be specified as one pair. So, make a setting as described below.
  Tandem axis: For a master axis, set an odd (1, 3, 5, 7, 9, ...) servo axis number. For a slave axis to be paired, set a value obtained by adding 1 to the value set for the master axis.
  EGB axis: For a slave axis, set an odd (1, 3, 5, 7, 9, ...) servo axis number. For a dummy axis to be paired, set a value obtained by adding 1 to the value set for the slave axis.

1260	The shift amount per one rotation of a rotary axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

Set the shift amount per one rotation of a rotary axis.
For the rotary axis used for cylindrical interpolation, set the standard value.

	#7	#6	#5	#4	#3	#2	#1	#0
1408					IRCx			

[Input type] Parameter input
[Data type] Bit axis

#3 IRCx The least input increment of the maximum cutting feedrates set in parameters Nos. 1430 and 1432 is:
0: Not multiplied by ten.
1: Multiplied by ten.

Set this parameter for the following axes, which are operated by the following functions:
- Spindle control axis by servo motor
- Tool rotary axis in the polygon turning function (T series)

If a rotation speed of 1000 (1/min) (=360000 (deg/min)) is to be used when this parameter is set to 1, set 36000.0 in parameter No. 1430/1432.

1430	Maximum cutting feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Specify the maximum cutting feedrate for each axis.

1432	Maximum cutting feedrate for all axes in the acceleration/deceleration before interpolation

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set a maximum cutting feedrate for each axis in the look-ahead acceleration/deceleration before interpolation mode such as AI contour control. When this parameter is set to 0 or the look-ahead acceleration/deceleration before interpolation mode is not set, the maximum cutting feedrate set in parameter No. 1430 is used.

1825	Servo loop gain for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01/sec
[Valid data range] 1 to 32767
Set the loop gain for position control for each axis.
When the machine performs linear and circular interpolation (cutting), the same value must be set for all axes. When the machine requires positioning only, the values set for the axes may differ from one another. As the loop gain increases, the response by position control is improved. A too large loop gain, however, makes the servo system unstable.
The relationship between the positioning deviation (the number of pulses counted by the error counter) and the feedrate is expressed as follows:

Positioning deviation = Feedrate / (60 × Loop gain)
Unit : Positioning deviation mm, inch or deg
Feedrate mm/min, inch/min, or deg/min
Loop gain 1/sec

1826	In-position width for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
The in-position width is set for each axis.
When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position. (The machine is in the in-position state.)

1827	In-position width in cutting feed for each axis

[Input type] Parameter input

[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set an in-position width for each axis in cutting feed. This parameter is used when bit 4 (CCI) of parameter No. 1801=1.

1828	Positioning deviation limit for each axis in movement

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.
If the positioning deviation exceeds the positioning deviation limit during movement, a servo alarm SV0411, "EXCESS ERROR (MOVING)" is generated, and operation is stopped immediately (as in emergency stop).
Generally, set the positioning deviation for rapid traverse plus some margin in this parameter. Refer to the following set value.

$$\textbf{Setting value} = \frac{\textbf{Rapid traverse rate}^{(\textbf{Note 1})}}{\textbf{60} \times \textbf{Servo loop gain}^{(\textbf{Note 2})}} \times \frac{1}{\textbf{Detection unit}} \times 1.2 \textbf{ to } 1.5$$

> ⚠ **CAUTION**
> If this parameter is not set correctly, machine or workpiece can be damaged.

> **NOTE**
> 1 Usually, this value is parameter No.1420. When the maximal feedrate of each axis exceeds parameter No.1420 according to the command and override, the value is the maximal feedrate of the each axis.
> 2 Usually, this value is parameter No.1825. When the servo loop gains other than parameter No.1825 is effective, the value is actual servo loop gains.

1829	Positioning deviation limit for each axis in the stopped state

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in the stopped state for each axis.
If, in the stopped state, the positioning deviation exceeds the positioning deviation limit set for stopped state, a servo alarm SV0410 is generated, and operation is stopped immediately (as in emergency stop).

	#7	#6	#5	#4	#3	#2	#1	#0
3702							EMS	

[Input type] Parameter input
[Data type] Bit path

#1 EMS The multi-spindle control is:
0: Used.

1: Not used.

	#7	#6	#5	#4	#3	#2	#1	#0
3703					MPP	MPM		2P2

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 2P2 When a multi-path system is used, inter-path spindle control allows:
0: Configuration where the spindle that belongs to one path only is shared between path 1 and path 2.
1: Configuration where the spindles that belong to path 1 and 2 are shared between the two paths.
When the spindle that belongs to an arbitrary path is shared between arbitrary paths, set bit 2 (MPM) of parameter No. 3703. (The meanings of signals used vary, so that ladder program modifications need to be made.)

#2 MPM When a multi-path system is used, the configuration allowed by inter-path spindle control:
0: Follows the setting of bit 0 (2P2) of parameter No. 3703.
1: Allows the sharing of the spindle that belongs to a path between arbitrary paths.

#3 MPP In multi-spindle control, a spindle selection using a programmed command instead of using the signals (SWS1 to SWS4<Gn027.0 to 2, Gn026.3>) is:
0: Not made.
1: Made.

> **NOTE**
> When this parameter is set to 1, set parameter No. 3781 at the same time.

	#7	#6	#5	#4	#3	#2	#1	#0
3706	TCW	CWM						

[Input type] Parameter input
[Data type] Bit path

#6 CWM
#7 TCW Set voltage polarity when the spindle speed is output from the following table

TCW	CWM	Voltage polarity
0	0	Both M03 and M04 positive
0	1	Both M03 and M04 negative
1	0	M03 positive, M04 negative
1	1	M03 negative, M04 positive

3717	Spindle amplifier number to each spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled axes

Set a spindle amplifier number to be assigned to each spindle.
0: No spindle amplifier is connected.
1: Spindle motor connected to amplifier number 1 is used.
2: Spindle motor connected to amplifier number 2 is used.
to
n: Spindle motor connected to amplifier number n is used.

3718	Subscript for display of a serial spindle (main spindle) or analog spindle

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 122

Set a subscript to be added to spindle speed display on a screen such as the position display screen.
Used when the spindle switching function is not used, or used for the main spindle when the spindle switching function is used.

> **NOTE**
> This parameter is invalid when an extended spindle name is used.

3719	Subscript for display of a serial spindle (sub-spindle)

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 122

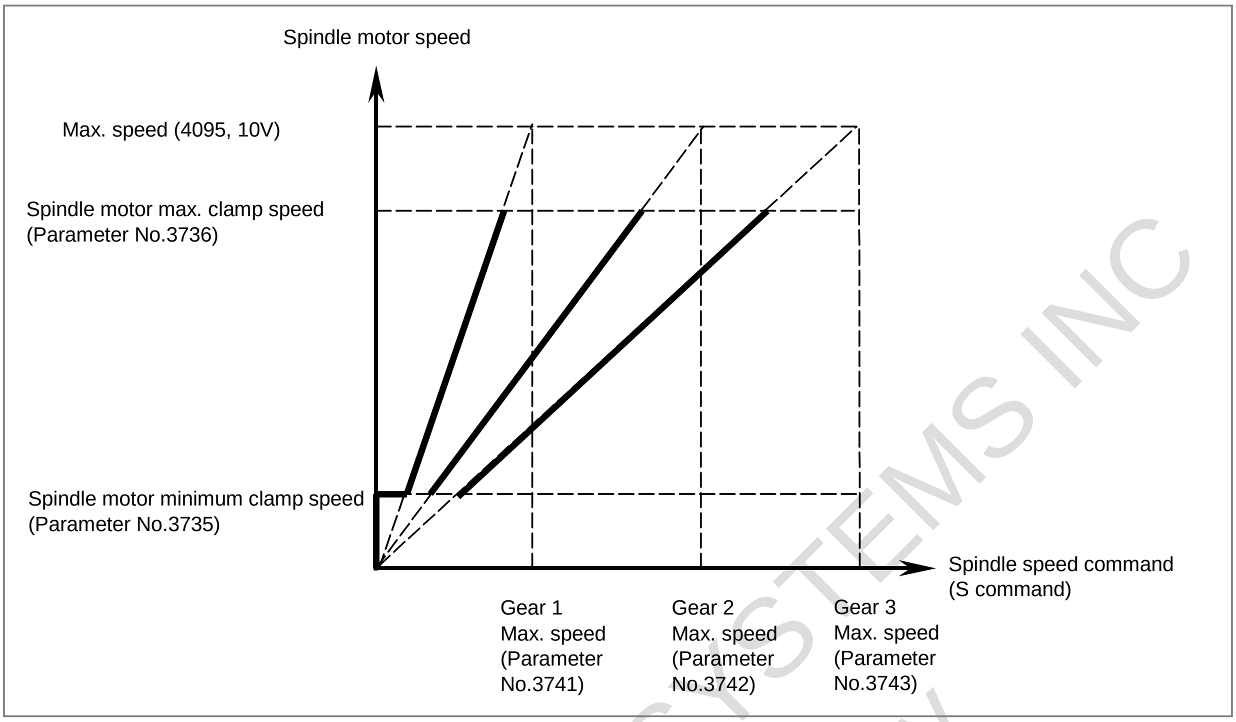
Set a subscript to be added to spindle speed display on a screen such as the position display screen.
Used for the sub-spindle when the spindle switching function is used.

> **NOTE**
> This parameter is invalid when an extended spindle name is used.

3741	Maximum spindle speed for gear 1
3742	Maximum spindle speed for gear 2
3743	Maximum spindle speed for gear 3
3744	Maximum spindle speed for gear 4

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

Set the maximum spindle speed corresponding to each gear.



3772	Maximum spindle speed

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

This parameter sets the maximum spindle speed.

When a command specifying a speed exceeding the maximum speed of the spindle is specified , or the speed of the spindle exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

> ⚠ **CAUTION**
> 1 When 0 is set in this parameter, the speed of the spindle is not clamped.
> 2 When spindle speed command control is applied using the PMC, this parameter has no effect, and the spindle speed is not clamped.

3775	Default P command value for spindle selection in multi-spindle control

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 32767

When bit 3 (MPP) of parameter No. 3703 is set to 1 and bit 2 (MPA) of parameter No. 3706 is set to 1 in multi-spindle control, set a default P command value applicable if S_P_ is not specified even once after power-up.

3781	P code for selecting the spindle in multi-spindle control

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767

If bit 3 (MPP) of parameter No. 3703 is set to 1, set the P code to select each spindle under multi-spindle control. Specify the P code in a block containing the S command.

[Example] If the P code value for selecting the second spindle is set to 3,
S1000 P3;
causes the second spindle to rotate at S1000.

> **NOTE**
> 1 This parameter is valid if bit 3 (MPP) of parameter No. 3703 is set to 1.
> 2 If this parameter is set to 0, the corresponding spindle cannot be selected by a P code.
> 3 Under multi-path control, the P code specified here is valid for each path.
> For instance, if the P code to select the first spindle of path 2 is set to 21, specifying S1000 P21; in path 1 causes the first spindle of path 2 to be rotated at S1000.
> 4 Identical P code values cannot be used for different spindles. (Identical P code values cannot be used even if the paths are different.)
> 5 When this parameter is used (when bit 3 (MPP) of parameter No. 3703 is set to 1), the spindle command selection signal is invalid.
> 6 To use this parameter, enable the multi-spindle control (bit 3 (MSP) of parameter No,8133 is 1).

	#7	#6	#5	#4	#3	#2	#1	#0
8003					FEX			

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter bit is set, the power must be turned off before operation is continued.

#3 FEX The maximum feedrate that can be achieved by the machine during cutting feed or continuous feed in PMC axis control, or spindle control with servo motor is:
0: Not extended.
1: Extended.

Restrictions
- Parameters for setting the time constants for linear acceleration/deceleration after interpolation and bell-shaped acceleration/deceleration after interpolation
When as the acceleration/deceleration type, linear acceleration/ deceleration after interpolation or bell-shaped acceleration/ deceleration after interpolation is used for each of rapid traverse, cutting feed, and manual feed, the maximum allowable time constant is a half of the maximum value that can be set conventionally.

The time constant parameters used are as follows:

Parameter No.	Meaning
1620	Time constant (T) used for linear acceleration/deceleration in rapid traverse for each axis, or time constant (T1) used for bell-shaped acceleration/deceleration in rapid traverse for each axis
1621	Time constant (T2) used for bell-shaped acceleration/deceleration in rapid traverse for each axis
1622	Time constant for acceleration/deceleration in cutting feed for each axis
1624	Time constant for acceleration/deceleration in jog feed for each axis
1626	Time constant for acceleration/deceleration in threading cycles for each axis
1769	Time constant for acceleration/deceleration after cutting feed interpolation in the mode of look-ahead acceleration/deceleration before interpolation
5271 to 5274	Time constant for acceleration/deceleration in rigid tapping extraction (first to fourth gears)
5365 to 5368	Time constant for bell-shaped acceleration/deceleration in rigid tapping (first to fourth gears)

- The waveform display function of VCMD on the SERVO GUIDE or the SERVO GUIDE Mate.
  As the feedrate increases, more data is acquired for VCMD waveform display, which can prevent waveforms from being displayed correctly.

> ⚠ **CAUTION**
> When this function is enabled, the feedrate is extended to the maximum value that can be specified for cutting feed or continuous feed in PMC axis control if CMR is 1. If CMR is greater than 1, the feedrate is limited to a value smaller than the maximum value that can be specified.
> Note that the maximum motor speed may be exceeded depending on the feedrate specified.

	#7	#6	#5	#4	#3	#2	#1	#0
11000	SRV						FSR	

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#1 FSR The axis to be subjected to servo motor-based spindle control is of a:
0: Semi-closed system.
1: Full-closed system.

#7 SRV Spindle control with servo motor are:
0: Not performed.
1: Performed

> **NOTE**
> When spindle control with servo motor are used for an axis, parameter No. 11010 must also be set for the axis.

	#7	#6	#5	#4	#3	#2	#1	#0
11001							TCR	

[Input type] Parameter input
[Data type] Bit axis

#1 TCR In SV speed control mode, time constant of acceleration/deceleration after interpolation for spindle control with servo motor is:
0: The parameter No. 1622.
(Time constant of acceleration/deceleration in cutting feed for each axis)
1: The parameter No. 11016.
(Time constant of acceleration/deceleration in SV speed control mode for each axis)
Set this parameter for the axis to be placed under spindle control with servo motor.

11010	Spindle number used by spindle control with servo motor or spindle control with Cs contour control

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to maximum number of controlled spindles
This parameter sets a spindle number for a servo axis for which spindle control with servo motor is performed.

> **NOTE**
> Set a spindle number for the axes set in bits 6 and 7 of parameter No. 11000. For axes for which spindle control with servo motor is not performed, set 0.

11011	Movement of spindle control with servo motor axis per revolution

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets a movement of axis per revolution for which spindle control with servo motor is performed.

11012	Spindle indexing speed for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

In spindle control with servo motor, set the spindle indexing speed for each axis.
If 0 is set, the spindle indexing speed is assumed to be the setting of parameter No. 11020 (speed (S_0) for switching acceleration/ deceleration for each axis).

11013	Positioning deviation limit for each axis in movement

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

This parameter sets the limit of positional deviation for each axis during movement in spindle control with servo motor.

11014	Positioning deviation limit for each axis in the stopped state

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

This parameter sets the limit of positional deviation at stop for each axis in spindle control with servo motor.

11015	Maximum motor speed

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

This parameter sets the maximum motor speed applicable when spindle control with servo motor is performed.

11016	Time constant of acceleration/deceleration in SV speed control mode for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

In spindle control with servo motor, set the time constant of acceleration/deceleration after interpolation in cutting feed for SV speed control mode. Set this parameter for the target axis for spindle control with servo motor. Set the time constant used for exponential acceleration/deceleration in cutting feed, bell-shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation in cutting feed for each axis. Type of acceleration/deceleration is applied by bits 0 (CTLx) and 1 (CTBx) of parameter No. 1610.

11017	FL rate of exponential acceleration/deceleration in SV speed control mode for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

In spindle control with servo motor, this parameter sets the lowest feedrate (FL rate) in exponential acceleration/deceleration for velocity control.

Set this parameter for the target axis for spindle control with servo motor.

11020	Acceleration/deceleration switching speed (S_0) for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

In spindle control with servo motor, this parameter sets the speed at which acceleration/ deceleration is changed to perform rotation control. (First step)

11021	Acceleration/deceleration switching speed (S_1) for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

In spindle control with servo motor, this parameter sets the speed at which acceleration/ deceleration is changed to perform rotation control. (Second step)

11030	Individual acceleration / deceleration 1 (Leg 1)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}/s
[Valid data range] 0 to 100000

In spindle control with servo motor, this parameter sets acceleration/deceleration to be applied to perform rotation control. When the speed ranges from 0 to acceleration switching speed 1, acceleration/deceleration 1 is applied. Acceleration switching speed 1 is the speed set in parameter No. 11020.

11031	Individual acceleration / deceleration 2 (Leg 2)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}/s
[Valid data range] 0 to 100000

In spindle control with servo motor, this parameter sets acceleration/deceleration to be applied to perform rotation control. When the speed ranges from acceleration switching speed 1 to acceleration switching speed 2, acceleration/ deceleration 2 is applied. Acceleration switching speed 1 and acceleration switching speed 2 are the speeds set in parameters Nos. 11020 and 11021, respectively.

11032	Individual acceleration / deceleration 3 (Leg 3)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}/s
[Valid data range] 0 to 100000

In spindle control with servo motor, this parameter sets acceleration/deceleration to be applied to perform rotation control. When the speed ranges from acceleration switching speed 2 to the maximum speed, acceleration/deceleration 3 is applied. Acceleration switching speed 2 is the speed set in parameter No. 11021.

10.20.2 Spindle Indexing Function

Format

G96.1 P_ R_ ;	After spindle indexing is completed, the operation of the next block is started.
G96.2 P_ R_ ;	Before spindle indexing is completed, the operation of the next block is started.
G96.3 P_ ;	After it is confirmed that spindle indexing is completed, the operation of the next block is started.

P: Spindle selection with multi-spindle control
R: Stoppage angle [deg] (0 to (parameter No. 1260))

To specify address P, use parameter No. 3781 (P code for selecting a spindle in multi-spindle control).
To turn the position control mode ON without performing spindle indexing, do so after canceling the SV speed control mode by issuing G96.1 command with no R specified when the motor is at a halt.
When the motor is rotating, issuing a G96.1 (or G96.2) command with no R specified results in the motor coming to a halt by behaving in the same manner as for R0.

NOTE
1 When using G96.2, issue G96.3 before another spindle move command, to make sure that the spindle is at a complete halt.
2 To issue G96.1, G96.2, or G96.3, use an independent block.

Spindle indexing command

- Move command

(1) Command waiting for spindle indexing to be completed
If G96.1 is issued, the next block is executed after spindle indexing is completed.
(2) Command not waiting for spindle indexing to be completed
If G96.2 is issued, the next block can be executed before spindle indexing is completed.

- Movement completion check command

G96.3 is used to check to see if spindle indexing is completed. If it has not been completed, the next block waits for spindle indexing to be completed. If it is completed, the next block is executed.

- SV speed control mode cancellation

If G96.1 is used to perform spindle indexing, the SV speed control mode is canceled when spindle indexing is completed.
If G96.2 is used to perform spindle indexing, G96.3 can be used to check to see if spindle indexing is completed and, if completed, cancel the SV speed control mode. Issuing G96.2 not followed by G96.3 cannot cancel the SV speed control mode even if spindle indexing is completed.
If SV speed control mode cancel, it is necessary to be commanded in the path to which the live tool axis belongs.

- Spindle indexing command during spindle rotation

Issuing G96.1 or G96.2 with a position specified during spindle rotation causes the spindle to stop at the specified position.
Example)
M03 S1000 ;...........................Rotation at S1000
G96.1 P1 R180.0 ;..................Stoppage of rotation at the 180° position

- Spindle indexing speed

Issuing G96.1 or G96.2 causes a move speed to be dedicated to spindle indexing. Specify the move speed for spindle indexing, using parameter No. 11012.

- Spindle indexing acceleration/deceleration

Fig. 10.20.2 (a) is the acceleration/deceleration specified by G96.1/G96.2.

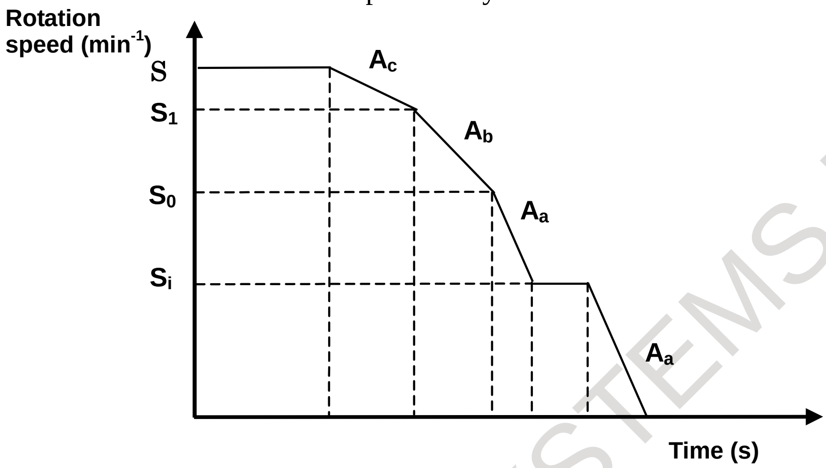
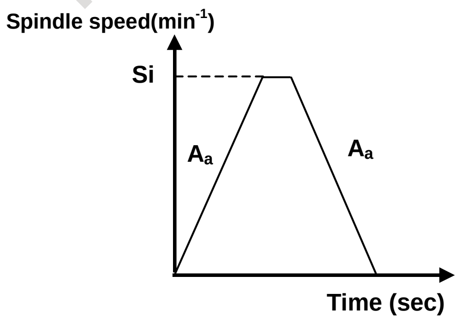


Fig. 10.20.2 (a)

S_0	Parameter No. 11020 setting (acceleration/deceleration is switched at a rotation speed of S_0 (min^{-1}).)
S_1	Parameter No. 11021 setting (acceleration/deceleration is switched at a rotation speed of S_1 (min^{-1}).)
S	Command-specified rotation speed (min^{-1})
Si	Spindle indexing speed (min^{-1}) parameter No. 11012
Aa	Parameter No. 11030 setting (acceleration/deceleration (min^{-1}/s) used between rotation speeds 0 and S_0 (span 1))
A_b	Parameter No. 11031 setting (acceleration/deceleration (min^{-1}/s) used between rotation speeds S_0 and S_1 (span 2))
Ac	Parameter No. 11032 setting (acceleration/deceleration (min^{-1}/s) used between rotation speeds S_1 and the maximum motor speed S (span 3))

Spindle indexing speed during a stop

If spindle indexing is executed (G96.1) while the rotation tool axis is stopped (with SV rotation control mode being ON), indexing is performed within one rotation, so that the spindle indexing speed (No. 11012) may not be attained.



Spindle indexing command (absolute coordinate/machine coordinate)

Using bit 0 (SIC) of parameter No. 11005 can select which coordinate system, absolute or machine, is to be used in spindle indexing.

Example:

If the difference between the machine and absolute coordinates (machine coordinate – absolute coordinate) is 100.000:

- Spindle indexing performed by turning the SV speed control mode signal SRVON<Gn521> OFF (“1” to “0”) or issuing a spindle indexing
  G code (with no R specified)
  1. If bit 0 (SIC) of parameter No. 11005 = 0:
     Spindle indexing is performed with a machine coordinate of 100.000 and an absolute coordinate of 0.000.
  2. If bit 0 (SIC) of parameter No. 11005 = 1:
     Spindle indexing is performed with a machine coordinate of 0.000 and an absolute coordinate of 260.000.
- G code-based spindle indexing (with R specified)
  Assuming that spindle indexing is specified with R100.000:
  1. If bit 0 (SIC) of parameter No. 11005 = 0:
     Spindle indexing is performed with a machine coordinate of 200.000 and an absolute coordinate of 100.000.
  2. If bit 0 (SIC) of parameter No. 11005 = 1:
     Spindle indexing is performed with a machine coordinate of 100.000 and an absolute coordinate of 0.000.

- Example program commands

1. A move command is issued to the spindle, using G96.2. The spindle starts moving, and the execution of the next block begins.
   The spindle keeps moving even when any other block is being executed.
   (The spindle indexing signal SPP<Fn522> is “1” during spindle indexing.)
2. When another command is issued to the spindle, G96.3 is used to heck in advance whether the spindle has finished moving.
   If the spindle is still moving (the spindle indexing signal is on), the CNC enters a wait state. If the spindle has finished moving, the command is issued to cause the spindle to start moving.

Example: Command not waiting for spindle indexing to finish and command checking whether spindle indexing has finished (parameter No. 3781 (S1) = 1)

Program command	Operation
G96.2 P1 R270.0 ;	Command not waiting for spindle indexing to finish. The first spindle S1 moves to 270.0.
G01 X10.0 Y20.0 F1000.0 ;	Starts cutting feed. No wait for spindle indexing to finish.
G02 X50.0 Y100.0 R50.0 ;	Starts circular interpolation. No wait for spindle indexing to finish.
G96.3 P1 ;	Checks whether spindle indexing has finished.
M29 S100 P1 ;	Starts rigid tapping if the spindle indexing signal is 0.
G84 X10.0 Y 20.0 R-5.0 Z-20.0	Waits for rigid tapping command if the spindle indexing signal is 1.

Notes

1. Spindle speed override signals and spindle stop signal are invalid from the cancellation of SV speed control mode to the completion of spindle indexing.
   - 1st spindle speed override signal SOV0 to SOV7<Gn030>
   - 2nd spindle speed override signal SOV20 to SOV27<Gn376>
   - 3rd spindle speed override signal SOV30 to SOV37<Gn377>
   - 4th spindle speed override signal SOV40 to SOV47<Gn378>
   - Spindle stop signal *SSTP<Gn029.6>

- Individual spindle stop signals *SSTP1, *SSTP2, *SSTP3, *SSTP4<Gn027.3, Gn027.4, Gn027.5>, <Gn026.6>

2. When the SV speed control mode is canceled, the spindle speed clamp is valid to the spindle speed command clamp (Function to spindle speed clamp by value set to internal relay address (R) of PMC specified by parameter No. 3773). Even if internal relay address (R) of PMC is changed from the cancellation of SV speed control mode to the completion of spindle indexing, the spindle speed clamp is not changed.
3. Specify an M code not involved in buffering as an M code to be used as a command for switching the SV speed control mode (parameters Nos. 3411 to 3420 and 11290 to 11299).
4. M code for switching the SV speed control mode must be specified in an independent block. When using multiple M commands in a single block, the M code must be specified as the first M command.
5. G96.1, G96.2, G96.3, G96.4, and M code for switching the SV speed control mode must be specified in the path to which the spindle of interest belongs.
6. After G96.2 command, command G96.3 before commanding move command such as rapid traverse (G00) or cutting feed (G01) to rotation axis. Move command is specified without commanding G96.3, alarm PS0445 “ILLEGAL AXIS OPERATION” is issued.
7. If G96.3 is not commanded after G96.2 command, SV speed control mode is not canceled. For example, reset before G96.2 command. Cancel the mode before issuing the axis move command.
8. If the spindle stops rotating because of G96.1, G96.2, the spindle speed command output becomes 0. To cause the spindle to restart rotating, place the spindle in the SV speed control mode, and then, issue an S command.
9. Spindle indexing is enabled only in the SV speed control mode. In any mode other than the SV speed control mode, use ordinary positioning.
10. If the specified spindle indexing speed is 0, the acceleration/deceleration switching rotation speed (stage 1) (parameter No. 11020) is used as the actual spindle indexing speed.
11. G96.1, G96.2, G96.3, G96.4 cannot be commanded for settings other than multi-spindle control type P.

Signal

Spindle indexing signal for each axis SPP1 to SPP8<Fn522>

[Classification] Output signal
[Function] These signals inform that spindle indexing for the respective axes is in progress.
[Output cond.] The signals are “1” if:
- Spindle indexing for the respective axes has not finished.

The signals are “0” if:
- Spindle indexing for the respective axes has finished.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn522	SPP8	SPP7	SPP6	SPP5	SPP4	SPP3	SPP2	SPP1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11005								SIC

[Input type] Parameter input
[Data type] Bit

#0 SIC Spindle indexing is:
0: Performed based on absolute coordinates.
1: Performed based on machine coordinates.

11012	Spindle indexing speed for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

In spindle control with servo motor, set the spindle indexing speed for each axis.
If 0 is set, the spindle indexing speed is assumed to be the setting of parameter No. 11020 (speed (S_0) for switching acceleration/ deceleration for each axis).

10.20.3 Speed-up of Spindle Indexing Function

Overview

In conventional spindle indexing, spindle indexing to the specified position after the speed was accelerated or was decelerated up to the spindle indexing speed was executed.
In this function, the spindle indexing is executed directly to the specified position. The spindle indexing time can be reduced.

Explanation

When bit 1 (HSP) of parameter No. 11002 is set to 1, spindle indexing when the spindle rotates is sped up.
When the spindle has stopped, the spindle indexing is the same as the conventional the spindle indexing.

Format

The command format for this type of the spindle indexing is the same as for the conventional type of the spindle indexing.
For details, see “Spindle Indexing” of the conventional type.

Speed-up of Spindle Indexing Function

In the following cases, the spindle indexing speed up is executed.
- G96.1(G96.2) is specified during spindle rotation
- SV speed control mode signals SRVON<Gn521> is set to ”0” during spindle rotation

In the following, the above-mentioned operation is described, "SV speed control mode off".

- Allowable spindle speed of spindle indexing (Parameter(No.11019))

Set the allowable spindle speed of the spindle indexing in parameter No. 11019.

(1) When the SV speed control mode is canceled, the spindle speed is higher than the allowable spindle speed. Decelerates from the spindle speed when the SV speed control mode is canceled, and the spindle indexing is executed to the specified position.

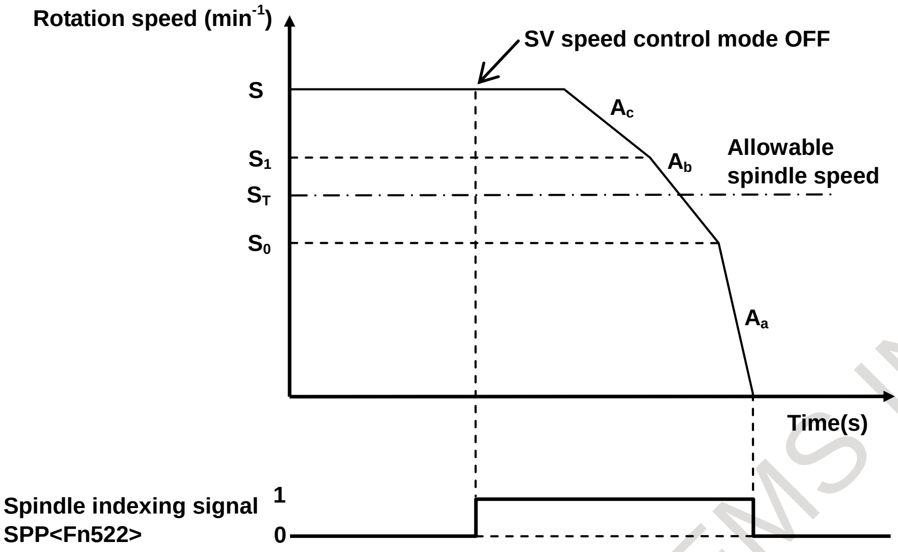


Fig. 10.20.3 (a) Spindle speed is higher than the allowable spindle speed

(2) When the SV speed control mode is canceled, the spindle speed is lower than the allowable spindle speed. The spindle accelerates once, and the spindle indexing is executed to the specified position.

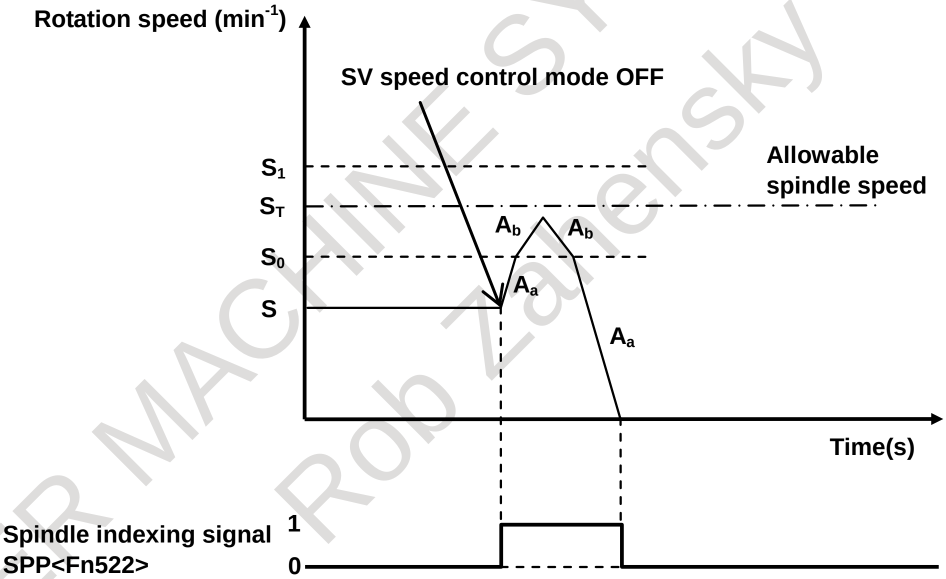


Fig. 10.20.3 (b) Spindle speed is lower than the allowable spindle speed

When accelerating during spindle indexing, the spindle speed is clamped at the lowest spindle speed among the following spindle speed.

- Parameter(No.11019) : Allowable spindle speed of spindle indexing
- Parameter(No.3772) : Maximum spindle speed
- Parameter(No.3741 to 3744) : Maximum spindle speed (Gear 1 to Gear 4)
- Spindle speed set to internal relay address (R signal) of PMC specified by parameter(No. 3773)

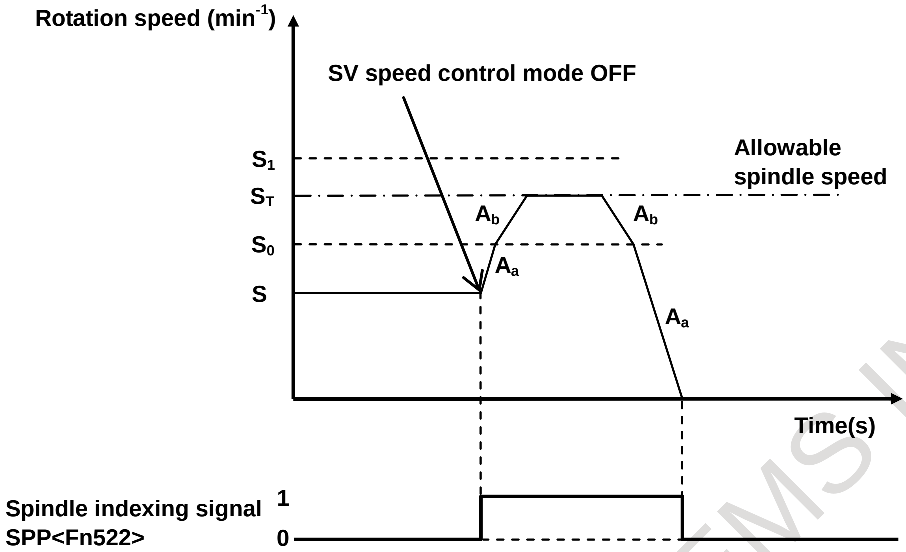


Fig. 10.20.3 (c) Clamped by allowable spindle speed

The spindle doesn't accelerate when the parameter (No.11019) is 0. The spindle decelerates from the spindle speed when the SV speed control mode is canceled, and the spindle indexing is executed to the specified position.

- Spindle indexing acceleration/deceleration

As for the spindle speed of the acceleration/deceleration switching and the acceleration after the SV speed control mode is canceled, the parameter in Table 10.20.3 is applied as shown in Fig. 10.20.3 (a) to Fig. 10.20.3 (c).

Table 10.20.3 Parameters for acceleration during spindle indexing

S_0	Acceleration switching speed (S_0) [min^{-1}] (parameter No.11020)
S_1	Acceleration switching speed (S_1) [min^{-1}] (parameter No.11021)
S	Specified rotation speed[min^{-1}]
S_T	Allowable spindle speed of spindle indexing[min^{-1}] (parameter No.11019)
A_a	Acceleration 1 [min^{-1}/s] (parameter No.11030)
A_b	Acceleration 2 [min^{-1}/s] (parameter No.11031)
A_c	Acceleration 3 [min^{-1}/s] (parameter No.11032)

- Spindle indexing signal for each axis

The spindle indexing signal SPP<Fn522> is “1” form Fig. 10.20.3 (a) to Fig. 10.20.3 (c) during spindle indexing until the spindle decelerates and stops after the SV speed control mode is canceled

Notes

1. Spindle speed override signals and spindle stop signal are invalid from the cancellation of SV speed control mode to the completion of spindle indexing.
   - 1st spindle speed override signal SOV0 to SOV7<Gn030>
   - 2nd spindle speed override signal SOV20 to SOV27<Gn376>
   - 3rd spindle speed override signal SOV30 to SOV37<Gn377>
   - 4th spindle speed override signal SOV40 to SOV47<Gn378>
   - Spindle stop signal *SSTP<Gn029.6>
   - Individual spindle stop signals *SSTP1, *SSTP2, *SSTP3, *SSTP4<Gn027.3, Gn027.4, Gn027.5>, <Gn026.6>
2. When the SV speed control mode is canceled, the spindle speed clamp is valid to the spindle speed command clamp (Function to spindle speed clamp by value set to internal relay address (R) of PMC specified by parameter No. 3773). Even if internal relay address (R) of PMC is changed from the

cancellation of SV speed control mode to the completion of spindle indexing, the spindle speed clamp is not changed.

3. Specify an M code not involved in buffering as an M code to be used as a command for switching the SV speed control mode (parameters Nos. 3411 to 3420 and 11290 to 11299).
4. M code for switching the SV speed control mode must be specified in an independent block. When using multiple M commands in a single block, the M code must be specified as the first M command.
5. G96.1, G96.2, G96.3, G96.4, and M code for switching the SV speed control mode must be specified in the path to which the spindle of interest belongs.
6. After G96.2 command, command G96.3 before commanding move command such as rapid traverse (G00) or cutting feed (G01) to rotation axis. Move command is specified without commanding G96.3, alarm PS0445 “ILLEGAL AXIS OPERATION” is issued.
7. If G96.3 is not commanded after G96.2 command, SV speed control mode is not canceled. For example, reset before G96.2 command. Cancel the mode before issuing the axis move command.
8. If the spindle stops rotating because of G96.1, G96.2, the spindle speed command output becomes 0. To cause the spindle to restart rotating, place the spindle in the SV speed control mode, and then, issue an S command.
9. Spindle indexing is enabled only in the SV speed control mode. In any mode other than the SV speed control mode, use ordinary positioning.
10. G96.1, G96.2, G96.3, G96.4 cannot be commanded for settings other than multi-spindle control type P.

Signal

Spindle indexing signal for each axis SPP1 to SPP8<Fn522>

[Classification] Output signal

[Function] These signals inform that spindle indexing for the respective axes is in progress.

[Output cond.] The signals are “1” if:

- Spindle indexing for the respective axes has not finished.

The signals are “0” if:

- Spindle indexing for the respective axes has finished.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn522	SPP8	SPP7	SPP6	SPP5	SPP4	SPP3	SPP2	SPP1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11002							HSP	

[Input type] Parameter input

[Data type] Bit axis

#1 HSP Speed-up of Spindle indexing is:

0: Disabled.

1: Enabled.

> **NOTE**
> This parameter is Automatic setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

11019	Allowable spindle speed of spindle indexing

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

When bit 1 (HSP) of parameter No. 11002 is 1, allowable spindle speed of speed-up spindle indexing is set.
When the SV speed control mode is canceled, the spindle speed is lower than the this parameter setting value, the spindle accelerates to this parameter setting value
The spindle doesn't accelerate when this parameter is 0. The spindle decelerates from the spindle speed when the SV speed control mode is canceled, and the spindle indexing is executed to the specified position.

10.20.4 Rigid Tapping with Servo Motor

Format

The command format for this type of rigid tapping is the same as for the conventional type of rigid tapping.
For details, see “Rigid Tapping” of the conventional type.

> **NOTE**
> Using rigid tapping commands requires putting the rotation axis servo motor in the position control mode in advance. If the servo motor is rotating, cause it to exit the SV speed control mode by issuing G96.1/G96.2. The SV speed control mode in-progress signal SVREV<Fn521> can be used to check what mode the servo motor rotation axis is in.

Rigid tapping specification

- **Feedrate**

For rigid tapping, the feedrate of a drilling axis is one specified in an F command, and that of the spindle is S × the amount of movement per live tool axis rotation (parameter No. 11011) [deg/min].
Feed per minute and feed per revolution are detailed later.
During rigid tapping, the spindle speed is limited by a parameter for specifying the maximum cutting feedrate for the axis used as a live tool axis, that is, parameter No. 1430 (or No. 1432 if look-ahead acceleration/ deceleration before interpolation is enabled).
Usually, the upper end of the maximum cutting feed range that can be specified by parameter No. 1430 or No. 1432 is 999999.999[deg/min] (equivalent to S2778 [min^{-1}]). For live tool axes specified for use in rigid tapping (bit 3 (IRC) of parameter No. 1408 = 1), the actual spindle speed is limited to within 10 times the setting of the maximum cutting feedrate parameter.
Example:
Maximum cutting feedrate parameter No. 1430 = 360000
Limit to the maximum spindle speed 360000×10 = 3600000 [deg/min] (S10000 [min^{-1}])

> ⚠ **CAUTION**
> Make the tapper thread pitch equal to one specified by the program (F, S). Otherwise, the tool or workpiece may be damaged.

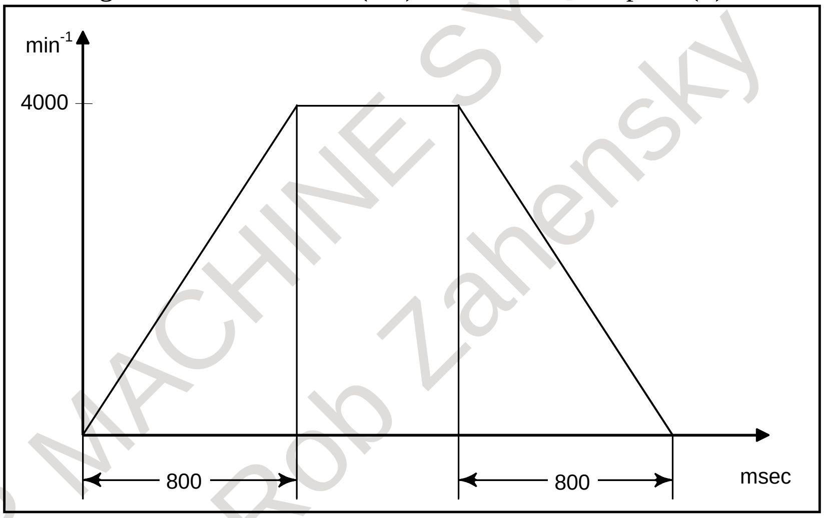
Acceleration/deceleration control

- Acceleration/deceleration after interpolation

In rigid tapping with servo motor, unlike the conventional type of rigid tapping, it is possible to apply linear acceleration/deceleration of constant acceleration/deceleration time type and bell-shape acceleration/deceleration.
Resetting bit 0 (SRBx) of parameter No. 11001 to 0 makes it possible to apply linear acceleration/deceleration after interpolation of constant acceleration time type. Setting the bit to 1 makes it possible to apply bell-shape acceleration/deceleration after interpolation of constant acceleration time type.
The time constant for each gear is specified in parameters Nos. 11060 to 11063. If bit 2 (TDR) of parameter No. 5201 = 1, the tool extraction time constant for each gear is specified in parameter Nos. 11065 to 11068. Specify each of these parameters for the live tool axis used in rigid tapping.
The acceleration/deceleration types and time constants used for drilling axes are set to the same values as for live tool axes.

NOTE
Time constants specified for rigid tapping with servo motor are different from those for the conventional type of rigid tapping.

Example: Parameter settings are: Time constant (TC) = 800 msec and speed (S) = 4000 min^{-1}



- Acceleration/deceleration before interpolation

If AI contour control I or II can be used in rigid tapping with servo motor, issuing a rigid tapping command in the "acceleration/deceleration before look-ahead interpolation" mode makes it possible to apply acceleration/deceleration before look-ahead interpolation to this type of rigid tapping. Acceleration/deceleration before look-head interpolation is effective when the AI contour control mode is ON. For details, see the Subsection of "AI Contour Control I and AI Contour Control II."
Parameter No. 11050 is used to specify the maximum allowable acceleration value for acceleration/deceleration interpolation used in rigid tapping. Parameter No. 11051 is used to specify the acceleration change time for look-ahead bell-shape acceleration/deceleration before interpolation. As for look-ahead acceleration/deceleration before interpolation used in rigid tapping, the maximum allowable acceleration value is 100000 (deg/s^2).
It is possible to change a speed of up to S1000 (min^{-1}) (equivalent to 360000 (deg/min) in 60 (ms).
The time constant (parameter No. 11052) for cutting feed acceleration/deceleration after interpolation usable in the "look-ahead acceleration/deceleration before interpolation" mode is a constant-time type.

> **NOTE**
> Specify the same time constant for both drilling and live tool axes. Otherwise, it is likely that the machine may malfunction.

- Feedrate clamp by parameter No.8465

In AI contour control, the maximum allowable feedrate is set by parameter No.8465. The maximum allowable feedrate of rigid tapping with servo motor during AI contour control is also set by parameter No.8465. If bit 0 (NCL) of parameter No.11003 is set to 1, the maximum allowable feedrate of rigid tapping with servo motor during AI contour control is not clamped by parameter No.8465. However, maximum allowable feedrate set by parameter No.1432 is available regardless of bit 0 (NCL) of parameter No.11003.

- Ignoring feedrate commands

When AI contour control II is enabled and bit 7 (NOF) of parameter No.8451 is set to 1, all feedrate commands are ignored and feedrate is specified by parameter No.8465. If bit 1 (FEN) of parameter No.11003 is set to 1, feedrate of rigid tapping with servo motor can be specified by machining program. But feedrate is limited by maximum allowable feedrate of parameter No.1432.

Reference position return

With this function, it is impossible to perform spindle orientation at the start of rigid tapping. Before issuing a rigid tapping command, perform spindle indexing or positioning at a point where drilling should begin. For details, see Subsection of "Spindle Indexing Function."

Diagnosis data

Also in rigid tapping with servo motor, information about rigid tapping is displayed on the diagnosis display screen. See Subsection of “Diagnosis Display” for what information is displayed.

Notes

If the pitch is very small or the amount of travel along the drilling axis is large, the amount of travel along the rotation axis becomes large, possibly resulting in alarm PS0003, “TOO MANY DIGITS”.

Parameter

1430	Maximum cutting feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Specify the maximum cutting feedrate for each axis.

1432	Maximum cutting feedrate for all axes in the acceleration/deceleration before interpolation

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set a maximum cutting feedrate for each axis in the look-ahead acceleration/deceleration before interpolation mode such as AI contour control. When this parameter is set to 0 or the look-ahead acceleration/deceleration before interpolation mode is not set, the maximum cutting feedrate set in parameter No. 1430 is used.

	#7	#6	#5	#4	#3	#2	#1	#0
8451	NOF							

[Input type] Setting input
[Data type] Bit path

#7 NOF In AI contour control, an F command is:
0: Not ignored.
1: Ignored.
When this parameter is set to 1, the specification of the maximum allowable feedrate set in parameter No. 8465 is assumed.

8465	Maximum allowable feedrate for AI contour control

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the maximum allowable feedrate for contour control.
If a feedrate higher than the setting of this parameter is specified in the AI contour control mode, the feedrate is clamped to that set in this parameter.
If this parameter is set to 0, no clamping is performed.
When bit 7 (NOF) of parameter No. 8451 is set to 1, the tool moves, assuming that the feedrate set in this parameter is specified. If 0 is set in this parameter at this time, a movement is made at the specified feedrate.

	#7	#6	#5	#4	#3	#2	#1	#0
11001								SRBx

[Input type] Parameter input
[Data type] Bit axis

#0 SRBx Acceleration/deceleration after interpolation in cutting feed during rigid tapping with servo motor is:
0: Linear acceleration/deceleration.
1: Bell-shaped acceleration/deceleration.

	#7	#6	#5	#4	#3	#2	#1	#0
11003							FEN	NCL

[Input type] Parameter input
[Data type] Bit path

#0 NCL In AI contour control, feedrate of rigid tapping with servo motor is:
0: limited by maximum allowable feedrate of parameter No.8465.
1: not limited by maximum allowable feedrate of parameter No.8465.
Maximum allowable feedrate of parameter No.1432 is enabled regardless of this parameter.

#1 FEN When ignoring feedrate commands is enabled (bit 7 (NOF) of parameter No.8451 is 1), feedrate of rigid tapping with servo motor is specified by:

0: parameter No.8465.

1: machining program.

Maximum allowable feedrate of parameter No.1432 is enabled regardless of this parameter.

11050	Maximum allowable acceleration rate in acceleration/deceleration before interpolation for each axis in rigid tapping

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, 0.0 to +10000.0)

Set a maximum allowable acceleration rate in look-ahead acceleration/ deceleration before interpolation for each axis.

If a value greater than 100000.0 is set, the value is clamped to 100000.0. If 0 is set, the specification of 100000.0 is assumed. If 0 is set for all axes, however, look-ahead acceleration/deceleration before interpolation is not performed.

11051	Acceleration change time of bell-shaped acceleration/deceleration before interpolation in rigid tapping

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 200

Set an acceleration change time of look-ahead bell-shaped acceleration/ deceleration before interpolation (time for changing from the state of constant federate (A) to the state of constant acceleration/deceleration (C) at the acceleration rate calculated from the acceleration rate set in parameter No. 11050: time of (B) in the Fig. 10.20.4 (d)).

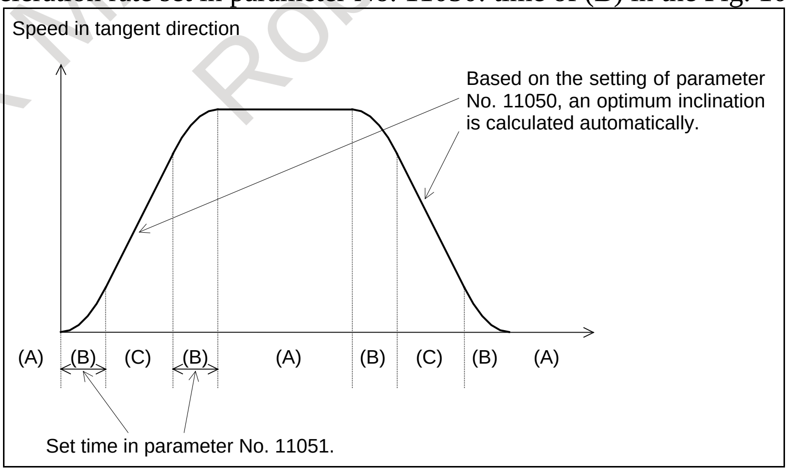


Fig. 10.20.4

11052	Time constant for acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode in rigid tapping

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec

[Valid data range] 0 to 4000

In the look-ahead acceleration/deceleration before interpolation mode as in AI contour control, not the ordinary time constant (parameter No. 1622) but the value of this parameter is used.

Be sure to specify the same time constant value for all axes except for a special application.

If different values are set, correct linear and circular figures cannot be obtained.

11060	**Time constant for acceleration/deceleration after cutting feed interpolation in rigid tapping (first gear)**
11061	**Time constant for acceleration/deceleration after cutting feed interpolation in rigid tapping (second gear)**
11062	**Time constant for acceleration/deceleration after cutting feed interpolation in rigid tapping (third gear)**
11063	**Time constant for acceleration/deceleration after cutting feed interpolation in rigid tapping (fourth gear)**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

For the time constants in rigid tapping with servo motors, parameters Nos. 11060 to 11063 are used, not parameters Nos. 5261 to 5264.

Set these parameters with a live tool axis in rigid tapping.

11065	**Time constant for acceleration/deceleration after cutting feed interpolation in rigid tapping extraction (first gear)**
11066	**Time constant for acceleration/deceleration after cutting feed interpolation in rigid tapping extraction (second gear)**
11067	**Time constant for acceleration/deceleration after cutting feed interpolation in rigid tapping extraction (third gear)**
11068	**Time constant for acceleration/deceleration after cutting feed interpolation in rigid tapping extraction (fourth gear)**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

If bit 2 (TDR) of parameter No. 5201 is set to 1, for the time constants in rigid tapping extraction with servo motors, parameters Nos. 11065 to 11068 are used, not parameters Nos. 5271 to 5274.

Set these parameters with a live tool axis in rigid tapping.

10.20.5 Threading, Feed per Revolution, and Constant Surface Speed Control

Overview

The following are supported in spindle control with servo motor:
- Feed per revolution
- Threading
- Constant surface speed control

- Feed per revolution

The feedrate for feed per revolution is obtained from the separate detector linked to the spindle. If the detector incorporated in the servo motor is used, the feedrate is calculated from the servo motor rotation speed and gear ratio.
Which detector to use is specified by bit 1 (OPTx) of parameter No. 1815.
See Subsection of "Feed Per Revolution/Manual Feed Per Revolution," for explanations about feed per revolution.

- Threading

Threading can be performed by reading the one-rotation signal from a separate detector linked to the spindle.
In a configuration in which the spindle and motor are linked at a specific gear ratio, a detector linked to the spindle at a gear ratio of 1:1 is necessary.
See Section of “THREADING,” for explanations about threading.

- Constant surface speed control

Using the S code (value following the letter S) to specify a surface speed (m/min or feet/min) enables the speed of the spindle to be controlled so that the surface speed is kept constant with regard to any change in the tool position.
As for gear change, use T type gear change.
See Section of “CONSTANT SURFACE SPEED CONTROL,” for explanations about constant surface speed control.

- Multi-spindle control

If the machine is equipped with a spindle controlled axis and spindle with servo motor, using rotation commands requires the multi-spindle control.
As for gear change, use T type gear change.
See Section of “MULTI-SPINDLE CONTROL,” for explanations about multi-spindle control.

Format

- Feed per revolution, threading, and constant surface speed control

The command format for feed per revolution, threading, and constant surface speed control explained above is the same as for the ordinary types of feed per revolution, threading, and constant surface speed control.

Examples

The following are examples in which the second spindle (C axis) is used for servo motor rotation control and position control.
(The G code system A in lathe system is selected.)

- Feed per revolution

Program command	Operation
M*** ;	The C axis rotation control mode is turned ON.
M03 S100 P2 ;	The C axis rotates at 100 min^{-1}.
G99 G01 Z-100. F10. ;	The Z axis moves at a feed-per-revolution speed of 1000 mm/min.
:	:
M*** ;	The C axis position control mode is set to ON (rotation control mode OFF). The spindle stops with C = 0.000.

- Threading

Program command	Operation
M*** ;	The C axis rotation control mode is set to ON.
M03 S100 P2 ;	The C axis rotates at 100 min^{-1}.
G32 Z-100.0 F1.0 ;	The Z-axis moves at a threading speed of 100 mm/min with a pitch of 1.000 mm.
:	:
M*** ;	The C axis position control mode is set to ON (rotation control mode OFF). The axis stops with C = 0.000.

- Constant surface speed control

Program command	Operation
M*** ;	The C axis rotation control mode is set to ON.
M03 S100 P2 ;	The C axis rotates at 100 min^{-1}.
G96 S12 P2.0 ;	The constant surface speed control mode is set to ON. The C axis rotates at 12 m/min.
G01 X100.0 Z20.0 ;	The tool moves in the constant surface speed control mode.
:	:
G97 S100 P2.0 ;	The constant surface speed control mode is set to OFF. The C axis rotates at 100 min^{-1}.
M*** ;	The C axis position control mode is set to ON (rotation control mode OFF). The axis stops with C = 0.000.

Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
1815							OPTx	

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 OPTx The separate position detector is:
0: Not to be used (semi-closed system)
1: To be used (full-closed system)

> **NOTE**
> In case of using the absolute position detector (bit 5 (APCx) of parameter No.1815 is set to 1), please set the following parameters that correspond to the absolute position detector which is actually used.
> - parameter No.1815#6, No.1815#0, No.1817#3, No.1868, No.2275#1, No.2394
>
> If these parameters are not set correctly, the machine coordinates are not correctly established at power-on.

2455	Integral part (α) of the number of pulses for one rotation

[Data type] Word axis
[Valid data range] 0 to 32767
If 0 is specified, no threading counter is output.
If the specified range is exceeded, an invalid parameter alarm SV0417 is issued.
(Detail number of diagnosis data No. 0352: 4553)

2456	Exponential part (β) of the number of pulses for one rotation

[Data type] Word axis
[Valid data range] 0 to 12
If the specified range is exceeded, an invalid parameter alarm SV0417 is issued.
(Detail number of diagnosis data No. 0352: 4563)

Number of pulses per detector rotation (Np) = $\alpha \times 2^{\beta}$
Determine α and β so that the above expression is satisfied.
For a closed system is in use, specify a value that corresponds to separate detector pulses.

Setting examples

System	Np(pulse/rev)	No.2455	No.2456	Remarks
αi Pulsecoder	Np=1,000,000	16384	7	
AB phase (12000p/rev)	Np=12,000	12000	0	
Serial output type rotary encoder (million pulses/rev)	Np=1,000,000	16384	7	
Serial output type rotary encoder (RCN223,723,220)	Np=8,000,000	16384	10	No.2275#0 No.2394 must be set.
5000λ/rev detector +High-resolution serial output circuit C or H	Np=5000×512 =5000×2^{9}	5000	9	No.2274#0=1 must be set.
18000λ/rev detector +High-resolution serial output circuit C or H	Np=18000×512 =18000×2^{9}	18000	9	No.2274#0=1 must be set.
36000λ/rev detector +High-resolution serial output circuit C or H	Np=36000×512 =18000×2^{10}	18000	10	No.2274#0=1 must be set.
5000λ/rev detector + analog SDU	Np=5000×512 =5000×2^{9}	5000	9	
18000λ/rev detector + analog SDU	Np=36000×512 =18000×2^{10}	18000	10	
36000λ/rev detector + analog SDU	Np=36000×512 =18000×2^{10}	18000	10	
2048λ/rev detector + Synchronous built-in servo motor position detection circuit	Np= 2,000,000	16384	8	

	#7	#6	#5	#4	#3	#2	#1	#0
11001						DDMx		

[Input type] Parameter input
[Data type] Bit axis

#2 DDMx The motor used for spindle control with servo motor is:
0: Not a DD motor.
1: A DD motor.

Notes

When using the functions explained above, enable the constant surface speed control (bit 0 (SSC) of parameter No,8133 is 1), spindle serial output (bit 5 (SSN) of parameter No,8133 is 0), and multi-spindle control (bit 3 (MSP) of parameter No,8133 is 1).

To use a DD motor, set bit 2 (DDMx) of parameter No. 11001 to 1.
Also needed are the CNC software and servo software supporting the functions.

10.20.6 Spindle Output Control with PMC

Overview

The “spindle control with servo motor” function enables the PMC to be sued for spindle output control.

How to specify

Once the SV speed control mode has been turned ON, spindle output control with PMC can be specified using the same commands as for ordinary spindle control.
Refer to Section of “SPINDLE OUTPUT CONTROL BY THE PMC,” of the connection manual for detailed descriptions.

NOTE
1 For the “spindle control with servo motor” function, the maximum motor speed is one specified in parameter No. 11015.
2 Even if using spindle output control by the PMC, specify parameters Nos. 3741 to 3744.

10.20.7 Speed Arrival Signals and Speed Zero Signals

Overview

In the function of spindle control with a servo motor, output signals equivalent to the speed arrival signals SARx and the speed zero signals SSTx for serial spindles are added. In the past, to determine whether the rotation axis with a servo motor reached at the specified speed, it was necessary to read the speed by using the real spindle speed output (T series) or PMC window function. By adding the speed arrival signals and the speed zero signals equivalent to those for serial spindles, the need is eliminated to create a new PMC ladder program for reading the servo motor speed and determining whether it has reached the specified speed. This makes it easy to replace the spindle (rotary tool) using a spindle motor with a servo motor.

Signal

speed arrival signals SVSAR1 to SVSAR8<Fn377>

[Classification] Output signal

[Function] In connection with the speed command in spindle control with a servo motor, these signals notify that the actual rotation speed of the servo motor has reached the previously set range.

[Output cond.] These signals are set to “1” in the following cases:

- In SV rotation control mode, the difference between the motor speed and the specified speed goes below the speed arrival detection level.
- The system enters position control mode.
- Spindle indexing is executed.
- Rigid tapping with a servo motor is executed.

These signals are set to “0” in the following cases:

- In SV rotation control mode, the difference between the motor speed and the specified speed goes above the speed arrival detection level.
- An emergency stop occurs. (Note)
- The system enters the servo off state.

> **NOTE**
> At the point when the servo software recognizes an emergency stop, SVSARx is set to “0”. Even if a condition for speed arrival is met, SVSARx is not set to “1”.

Speed zero signals SVSST1 to SVSST8<Fn376>

[Classification] Output signal

[Function] These signals notify that the rotation speed of the servo motor used for spindle control with a servo motor has become equal to or less than the speed detection level.

[Output cond.] These signals are set to 1 in the following case:

- The motor speed goes below the speed zero detection level.

These signals are set to 0 in the following case:

- The motor speed goes above the speed zero detection level.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn376	SVSST8	SVSST7	SVSST6	SVSST5	SVSST4	SVSST3	SVSST2	SVSST1
Fn377	SVSAR8	SVSAR7	SVSAR6	SVSAR5	SVSAR4	SVSAR3	SVSAR2	SVSAR1

Parameter

2482	Speed arrival detection level

[Input type] Parameter input

[Data type] Word axis

[Unit of data] 0.1%

[Valid data range] 0 to 1000

[Recommended value] 0 (15%)

This parameter sets the detection range for the speed arrival signal SVSARx as a percentage of the specified speed. The actual detection level used is the setting of this parameter plus an offset of 50/min.

If the difference between the motor speed and the specified speed reaches the speed arrival detection level, the speed arrival signal SVSARx is set to 1.

2483	Speed zero detection level

[Input type] Parameter input
[Data type] Word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 32767
[Recommended value] 0 (45/min)

This parameter sets the detection range for the speed zero signal SVSSTx.
If the motor speed becomes equal to or less than the speed zero detection level, the speed zero signal SVSSTx is set to 1.

Note

- This function does not support composite control.
- Servo software supporting this function is required.

10.20.8 Using Speed Control to Improve Spindle Control with Servo Motor

Overview

Conventionally, spindle control with servo motor has used position control for the SV rotation control mode. This function enables speed control to be used.

Specification

Setting bit 0 (SPCx) of parameter No. 11000 to 1 enables speed control to be used for the SV rotation control mode. (If bit 0 (SPCx) of parameter No. 11000 is 0, position control is used for the SV rotation control mode.)

To turn the SV rotation control mode ON, use a G code (G96.4) or the signal SRVON (Gn521) = 1 in the same manner as conventional.
To turn the SV rotation control mode OFF, use a G code (G96.1) or the signal SRVON (Gn521) = 0.

If speed control is used for the SV rotation control mode, issuing G96.1 during rotation outputs S0, causing the SV rotation control mode to be turned OFF after rotation stops. In this case, however, spindle indexing does not operate.

When using SRVON (Gn521) = 0 to turn the SV rotation control mode OFF, previously stop rotation by issuing S0 or resetting the spindle stop signal (SSTP (Gn029.6) or SSTPx (Gn027.3, 4, 5, Gn026.6)) to 0. Resetting the signal SRVON (Gn521) to 0 during rotation cannot turn the SV rotation control mode OFF.

If speed control is used for the SV rotation control mode, it is impossible to use spindle indexing (G96.1/G96.2/G96.3). When positioning the spindle at a specified location, previously turn the SV rotation control mode OFF.

Even when this function is used, positional information is updated during rotation. The current position managed by the CNC is updated by follow-up. The value of position error displayed at diagnosis data No.300 etc. is the difference between actual machine position and current position updated by follow-up.

NOTE

Status output signals in servo rotation control mode (SV rotation control mode) are as follows.

Status output signals in servo rotation control mode

Status output signals	Position control (Bit 0 (SPC) of parameter No.11000 is 0)	Speed control (Bit 0 (SPC) of parameter No.11000 is 1)	
		Bit 1 (VCSx) of parameter No.11004 is 0	Bit 1 (VCSx) of parameter No.11004 is 1
Axis moving signals MV1 to MV8<Fn102>	Spindle rotation : “1” Spindle stop : “0”	“0” at all times	Velocity command 0 : “0” Velocity command except for 0 : “1”
Axis moving direction signals MVD1 to MVD8<Fn106>	Spindle forward rotation : “0” Spindle reverse rotation : “1”	The status when servo rotation control mode is changed is maintained.	Positive velocity command : “0” Negative velocity command : “1”
In-position signals INP1 to INP8<Fn104>	In-position : “1” Not in-position : “0”	“1” at all times	“0” at all times

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11000	**SRVx**							**SPCx**

[Input type] Parameter input
[Data type] Bit axis

NOTE

When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 SPCx In spindle control with servo motor, the SV rotation control mode uses:
0: Position control.
1: Speed control.

#7 SRVx Spindle control with servo motor are:
0: Not performed.
1: Performed

	#7	#6	#5	#4	#3	#2	#1	#0
11004							**VCSx**	

[Input type] Parameter input
[Data type] Bit axis

#1 VCSx When SV rotation control mode of spindle control with servo motor is speed control (bit 0 (SPCx) of parameter No.11000 is 1), improvement of status output signals is:
0: Disabled.
1: Enabled.

Specifications of status output signals are as follows.

Status output signals	Bit 1 (VCSx) of parameter No.11004 is 0	Bit 1 (VCSx) of parameter No.11004 is 1
Axis moving signals MV1 to MV8<Fn102>	"0" at all times	Velocity command 0: "0" Velocity command except for 0: "1"
Axis moving direction signals MVD1 to MVD8<Fn106>	The status when servo rotation control mode is changed is maintained.	Positive velocity command: "0" Negative velocity command: "1"
In-position signals INP1 to INP8<Fn104>	"1" at all times	"0" at all times

10.20.9 Spindle Synchronous Control for Spindle Control with Servo Motor

Overview

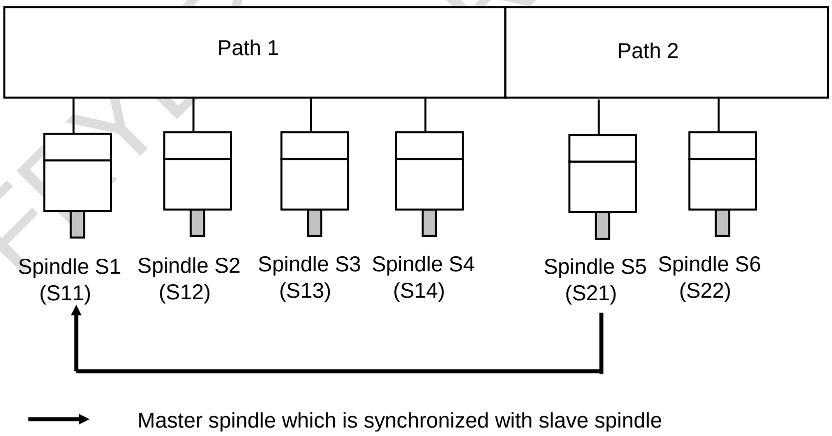
This function enables control spindles synchronously in spindle control using servo motor. It also enables the control of the rotation phase of a spindle, a polygon-bar as well as a round-bar can be exchanged between two spindles. A combination of a master spindle and slave spindle in spindle synchronous control can be selected arbitrarily.
This function can be used with servo motors in the state of SV speed control mode.

Specification

Synchronous-spindle configuration

In spindle synchronous control, the spindle which is issued S command is called the master spindle. A spindle which ignores any S command that is issued for it, and instead rotates synchronously with the master spindle, is called the slave spindle.
Example :
Combining spindles of different paths for spindle synchronization
Parameter setting
Bit 4 (SSS) of parameter No.3704 (PATH1)=1, (PATH2)=1
Parameter No.4831(ALL) =0
Parameter No.4832(S1) =0, (S2) =0,(S3) =0,(S4) =0,(S5) =1,(S6) =0

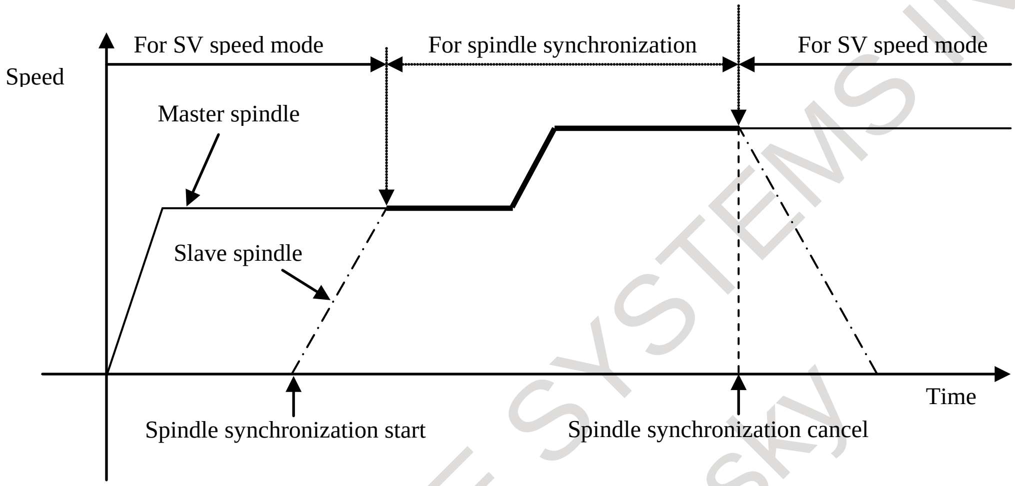
Master spindle	Slave spindle
First spindle of path 1(S11)	First spindle of path 2(S21)



Acceleration/deceleration

As for the acceleration/deceleration when the slave axis synchronizes with the master axis, the parameters Nos.11020 to 11021 and Nos.11030 to 11032 at the acceleration/deceleration of spindle control function with servo motor or the parameters Nos.11070 to 11071 and Nos.11080 to 11082 at the acceleration of spindle synchronous control for spindle control with servo motor becomes effective. In case of bit 6 (DCSx) of parameter No.11001 for the acceleration/deceleration is set to 1, the parameter(Nos.25720 to 25721 and Nos.25730 to 25732) at the deceleration of spindle synchronous control for spindle control becomes effective.

Ex.) In case of bit 6 (CSA) of parameter No.11005 =1:



Speed synchronization

When the spindle synchronous control signal is set to 1, the spindle synchronous control mode is set. When spindle synchronous control is specified, slave spindle is accelerated or decelerated to a specified speed of master spindle then enters the synchronous control state.
The spindle synchronous speed control completion signal is output when the two spindles have reached the speed matching a specified spindle synchronous speed and the speed difference between the two spindles is
within the value set in parameter No.11042.
The S command to the master spindle is clamped by the maximum speed of the slave spindle if S command is more than the maximum speed of the slave spindle(No.3741-No.3744).

Phase matching

Spindle phase synchronization is executed by setting the spindle phase synchronization control signal to set to 1 in the spindle synchronous control mode (after the output of the spindle synchronous speed control completion signal). The synchronous state between the two axes is not maintained during spindle phase synchronization operation (until the spindle phase synchronization control completion signal is set to 1).
The spindle phase synchronization completion signal is output when the error difference between the two spindles lies within the allowable number of pulses set in the CNC parameter No.4810. When the two spindles hold a workpiece, do not specify spindle phase synchronization.
When spindle phase synchronization control is executed, the speed fluctuation of slave spindle can be reduced. When parameter No.11040 is 0, the speed fluctuation of the slave spindle increases because the amount of the movement of the slave spindle is output at a time and the position deflection increases rapidly. The phase match can be executed smoothly because the amount of the movement of the slave spindle is divided by the number of pulses of parameters No.11040 of each 8msec.
When spindle phase synchronization control is specified, phase matching is executed by movement of slave spindle. The operation of the master spindle is not influenced. Therefore, you can command spindle phase synchronization control at the same time by each slave spindle.

Speed command

In the spindle synchronous control mode, the speed command for the master spindle is valid.
In the spindle synchronous control mode, a speed for the master spindle can be command in the same way as for an ordinary S command.
When 0 is set in the parameter corresponding to a selected gear, the spindle does not make a synchronous rotation.
A S command for a master spindle before the spindle synchronous control mode is set are valid even in the spindle synchronous control mode. A speed command for a master spindle during spindle synchronous control is also valid as a command for the master spindle after spindle synchronous control is canceled.

Command by PMC DI signals

Spindle synchronous control can be commanded by the DI signal of PMC.

- Spindle synchronous control mode ON

Spindle synchronous control is executed when spindle synchronous control signal of each spindle SPSYCs<Gn288.0~3> is set to 1. Phase match is executed when spindle phase synchronization control signal of each spindle SPPHSs<Gn289.0~3> is set to 1.

- Spindle synchronous control mode OFF

Spindle synchronous control is released when spindle synchronous control signal of each spindle SPSYCs<Gn288.0~3> is set to 0.

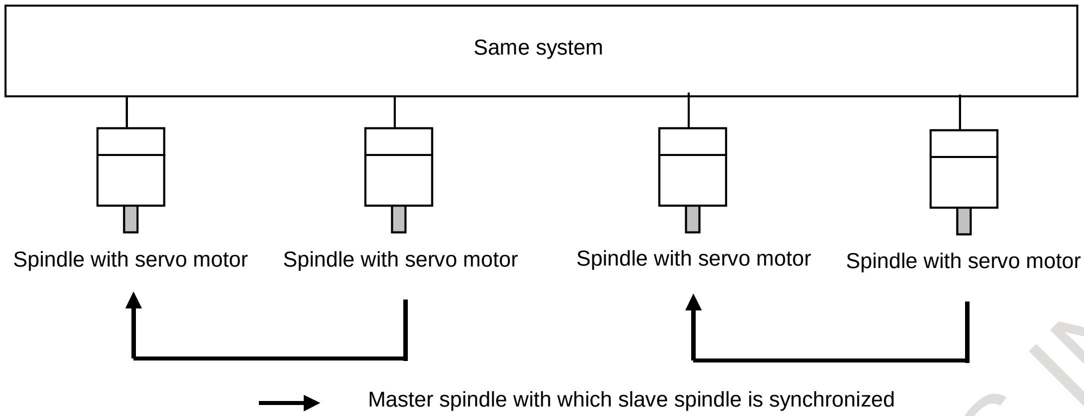
Supplement

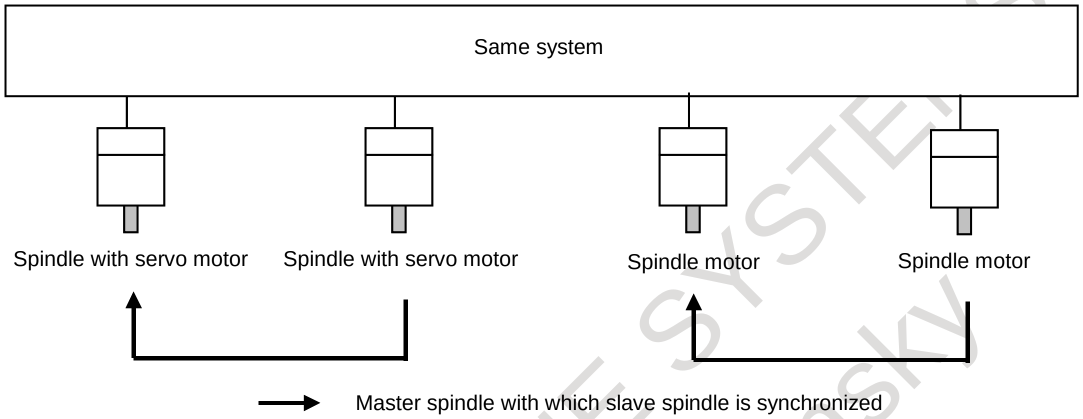
To monitor an error mutually between two spindles in the spindle synchronous control mode, the phase error monitor signal SYCAL<Fn044.4>(SYCAL1 to SYCAL4<Fn043.0> to <Fn043.3>) is available. A synchronization error between two spindles is always monitored. When a synchronization error exceeding the value (absolute number of error pulses) set in parameter No.4811 is detected, this signal is set to “1”. When the synchronization error does not exceed the value set in parameter No.4811, the signal is set to “0”.
Even when two spindles hold a workpiece in the synchronous control state, constant surface speed control can be performed. Even if the speed of a spindle changes, however, the synchronous speed changes within the parameter-set range (acceleration/deceleration time constant in spindle synchronous control).
In spindle synchronous control, the spindle speed offset value (parameter No.3731) is invalid.
The slave axis moves whenever instructing in the spindle phase synchronization.

Limitation

When both a master spindle and a slave spindle are SV speed control mode, you can command to change spindle synchronous mode.
For spindles during the spindle synchronous control mode, commands for rigid tapping, simple synchronous control, and so on cannot be specified.
The ratio of the gear of the master axis and the slave axis should be the same.
If SV speed control mode of spindle control with servo motor is velocity control type (bit 0 (SPCx) of parameter No.11000 =1), this function cannot be used.

The example that can be used is shown as follows. The pair that can be used is shown in the example. if each pair is like a pair of “synchronization with spindle control with servo motor or a pair of the usual synchronization with spindle control”, two or more pairs of the synchronization can be specified.





Diagnosis data (Spindle synchronous control)

300	Position deviation of each servo motor

The position deviation of each servo motor is indicated.

425	Synchronization error of each spindle

The absolute value of the synchronization error of each spindle in spindle synchronization is indicated. A synchronization error is indicated with a slave spindle.

Signal

Spindle synchronous control signal of each spindle
SPSYC1 to SPSYC4<Gn288.0 to Gn288.3>

[Classification] Input signal

[Function] These signals turn on the spindle synchronous control mode.

[Operation] When each of these signals is set to “1”, the spindle synchronous control mode with a spindle set as a slave spindle is set.
When each of these signals is set to “0”, the spindle synchronous control mode with a spindle set as a slave spindle is canceled. When SPSYCs is set to “1”, spindle synchronization with a spindle set as a slave spindle is performed. These signals are valid only when bit 4 (SSS) of parameter No.3704 is set to 1 and bit 5 (SCB) of parameter No.4800 is set to 0.
Which spindle is synchronized with which spindle is set using parameter No.4831 or No.4832.

Spindle phase synchronization control signal of each spindle
SPPHS1 to SPPHS4 <Gn289.0 to Gn289.3>

[Classification] Input signal

[Function] These signals turn on the spindle phase synchronization control mode (phase matching) for each spindle.

[Operation] When each of these signals is set to “1”, spindle phase matching is performed in the spindle synchronous control mode with a spindle set as a slave spindle. These signals are valid when the spindle synchronous control signal SPSYCs for each spindle is set to “1”. Specify each of these signals after the spindle synchronous speed control completion signal FSPSYs for each spindle is set to “1”.
A spindle phase synchronization control operation is started on the rising edge of each of these signals. So, phase matching once performed is not lost when each of these signals is set to “0”. These signals are valid only when bit 4 (SSS) of parameter No.3704 is set to 1 and bit 5 (SCB) of parameter No.4800 is set to 0. Set a phase shift amount in parameter No.11041.

> **NOTE**
> Before specifying these signals, ensure that the spindle synchronous speed control completion signal FSPSYs is set to “1”. When the spindle synchronous speed control completion signal FSPSYs is set to “0”, phase matching operation is not performed.

Spindle synchronous speed control completion signal for each spindle
FSPSY1 to FSPSY4<Fn288.0 to Fn288.3>

[Classification] Output signal

[Function] These signals show that spindle synchronous control (speed synchronization) with each spindle set as a slave spindle is completed.

[Output cond.] This signal is set to “1” in the following case:

- This signal is output when two spindles have reached the speed equivalent to a specified spindle synchronous speed and the speed difference between the two spindles is equal to or less than the value set in parameter No.11042 in the spindle synchronous control mode.

This signal is set to “0” in the following cases:

- In the spindle synchronous control mode, two spindles have not reached the speed equivalent to a specified spindle synchronous speed.
- In the spindle synchronous control mode, the speed difference between two spindles is greater than the value set in parameter No.11042.
- The spindle synchronization control mode is not set.
  These signals are valid only when bit 4 (SSS) of parameter No.3704 is set to 1 and bit 5 (SCB) of parameter No.4800 is set to 0.

> **NOTE**
> Even if this signal is once set to “1”, this signal is set to “0” when the speed difference exceeds the value set in parameter No.11042 for a cause such as cutting load fluctuation.

Spindle phase synchronization control completion signal of each spindle
FSPPH1 to FSPPH4<Fn289.0 to Fn289.3>

[Classification] Output signal

[Function] These signals show that spindle phase synchronization control (phase matching) with each spindle set as a slave spindle is completed.

[Output cond.] This signal is set to “1” in the following case:

- This signal is output when phase matching is completed with the spindle phase synchronization control signal (when the error pulse difference between the two spindles is equal to or less than the value set in parameter No.4810) after the two spindles have reached the speed equivalent to a specified spindle synchronous speed in the spindle synchronous control mode.

This signal is set to “0” in the following cases:

- In the spindle synchronous control mode, the phase matching of two spindles is not completed.
- In the spindle synchronous control mode, the error pulse difference between two spindles is greater than the value set in parameter No. 4810.
- The spindle synchronous control mode is not set.
- The spindle phase synchronization control mode is not set.

  These signals are valid only when bit 4 (SSS) of parameter No.3704 is set to1 and bit 5 (SCB) of parameter No.4800 is set to 0.

> **NOTE**
> Even if this signal is once set to “1”, this signal is set to “0” when the speed difference exceeds the value set in parameter No.4810 for a cause such as cutting load fluctuation.

Spindle phase error monitor signal for each spindle
SYCAL1 to SYCAL4<Fn043.0 to Fn043.3>

[Classification] Output signal

[Function] These signals show that in the spindle synchronous control mode with each spindle set as a slave spindle, the error pulse difference between two spindles is greater than a parameter-set value.

[Output cond.] This signal is set to “1” in the following case:

- In the spindle synchronous control mode, the error pulse difference between two spindles is greater than the value set in parameter No.4811 after spindle synchronous control is completed.

This signal is set to “0” in the following cases:

- The spindle synchronous control mode is not set.
- In the spindle synchronous control mode, the error pulse difference between two spindles is equal to or less than the value set in parameter No.4811.

  These signals are valid only when bit 4 (SSS) of parameter No.3704 is set to 1 and bit 5 (SCB) of parameter No.4800 is set to 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn288					SPSYC4	SPSYC3	SPSYC2	SPSYC1

	#7	#6	#5	#4	#3	#2	#1	#0
Gn289					SPPHS4	SPPHS3	SPPHS2	SPPHS1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn043					SYCAL4	SYCAL3	SYCAL2	SYCAL1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn288					FSPSY4	FSPSY3	FSPSY2	FSPSY1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn289					FSPPH4	FSPPH3	FSPPH2	FSPPH1

Parameter

Parameters which are used for this function are as follows.

	#7	#6	#5	#4	#3	#2	#1	#0
11000								SPCx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 SPCx SV speed control mode of spindle control with servo motor is:
0: Position control type.
1: Velocity control type.

> **NOTE**
> Spindle synchronous control for spindle control with servo motor (bit 3 (SSY) of parameter No.11005=1) is only available with this parameter setting 0.

	#7	#6	#5	#4	#3	#2	#1	#0
11001		DCSx						

[Input type] Parameter input
[Data type] Bit axis

#6 DCSx Acceleration/deceleration for deceleration is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
11005		CSA			SSY		CSC	

[Input type] Parameter input
[Data type] Bit

#1 CSC The coordinate of spindle phase synchronization control servo motor is:
0: Absolute coordinate.
1: Machine coordinate.

#3 SSY Spindle synchronous control with servo motor is:
0: Disabled.
1: Enabled.

> **NOTE**
> This function is only available with the parameter setting of bit 0 (SPCx) of parameter No.11000=0.

#6 CSA Acceleration/deceleration for spindle synchronous control with servo motor is:
0: Acceleration/deceleration for speed control.
1: Acceleration/deceleration for spindle synchronous control.

11040	Spindle phase synchronization with servo motor compensation data

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] pulse
[Valid data range] 0 to 359999

This parameter reduces speed fluctuations in case of aligning phase of spindle in spindle phase synchronization control.
When this parameter is 0, since the phase alignment amount is output at a time, the position deviation quickly becomes large, and there are large speed fluctuation on phase alignment.
It is possible to perform smooth phase alignments as phase alignment amounts is separated by the number of 8 msec pulses set in this parameter.

11041	Shift amount for spindle phase synchronization control with servo motor

[Input type] Parameter input
[Data type] Real spindle
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 to 359.999

Sets the shift amount of slave spindle from master spindle at spindle phase synchronization control with servo motor.

11042	Detection level for spindle synchronization with servo motor completion signal

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 32767

For the synchronization speed command at synchronization control with servo motor, if the deviations of the respective spindle motor speeds are within the setting level, the spindle synchronization control complete signal(FSPSY) becomes 1.

11070	Acceleration/deceleration switching speed (S_0) of spindle synchronous

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 999999999

Set the acceleration/deceleration switching points S_0 for spindle synchronous control with servo motor.

11071	Acceleration/deceleration switching speed (S_1) of spindle synchronous

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 999999999

Set the acceleration/deceleration switching points S_1 for spindle synchronous control with servo motor.

11080	Individual acceleration / deceleration 1 of spindle synchronous (Leg 1)

[Input type] Parameter input

[Data type] 2-word axis
[Unit of data] min^{-1}/s
[Valid data range] 0 to 100000

In spindle control with servo motor, this parameter sets acceleration/deceleration to be applied to perform spindle synchronous control. When the speed ranges from 0 to acceleration switching speed 1, acceleration/deceleration 1 is applied. Acceleration switching speed 1 is the speed set in parameter No. 11070.

11081	Individual acceleration / deceleration 2 of spindle synchronous (Leg 2)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}/s
[Valid data range] 0 to 100000

In spindle control with servo motor, this parameter sets acceleration/deceleration to be applied to perform spindle synchronous control. When the speed ranges from acceleration switching speed 1 to acceleration switching speed 2, acceleration/ deceleration 2 is applied. Acceleration switching speed 1 and acceleration switching speed 2 are the speeds set in parameter Nos. 11070 and 11071, respectively.

11082	Individual acceleration / deceleration 3 of spindle synchronous (Leg 3)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}/s
[Valid data range] 0 to 100000

In spindle control with servo motor, this parameter sets acceleration/deceleration to be applied to perform spindle synchronous control. When the speed ranges from acceleration switching speed 2 to the maximum speed, Acceleration/deceleration 3 is applied. Acceleration switching speed 2 is the speed set in parameter No. 11071.

25720	Acceleration/deceleration switching speed (S_{10}) of spindle synchronous for deceleration

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 999999999

Set the acceleration/deceleration switching points of spindle synchronous control with servo motor for deceleration S_{10} .

> **NOTE**
> When bit 6 (DCSx) of parameter No.11001 and bit 6 (CSA) of parameter No.11005 are set to 1, this parameter becomes effective.

25721	Acceleration/deceleration switching speed (S_{11}) of spindle synchronous for deceleration

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 999999999

Set the acceleration/deceleration switching points of spindle synchronous control with servo motor for deceleration S_{11}.

> **NOTE**
> When bit 6 (DCSx) of parameter No.11001 and bit 6 (CSA) of parameter No.11005 are set to 1, this parameter becomes effective.

25730	Individual acceleration / deceleration of spindle synchronous for deceleration (0 to S_{10})

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}/s
[Valid data range] 0 to 100000

Set the acceleration/deceleration of spindle synchronous control with servo motor for deceleration of Leg 1 (0 to S_{10}).
Acceleration switching speed 1 is the speed set in parameter No. 25720.

> **NOTE**
> When bit 6 (DCSx) of parameter No.11001 and bit 6 (CSA) of parameter No.11005 are set to 1, this parameter becomes effective.

25731	Individual acceleration / deceleration of spindle synchronous for deceleration (S_{10} to S_{11})

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}/s
[Valid data range] 0 to 100000

Set the acceleration/deceleration of spindle synchronous control with servo motor for deceleration of Leg 2 (S10 to S11). Acceleration switching speed 1 and acceleration switching speed 2 are the speeds set in parameter Nos. 25720 and 25721, respectively.

> **NOTE**
> When bit 6 (DCSx) of parameter No.11001 and bit 6 (CSA) of parameter No.11005 are set to 1, this parameter becomes effective.

25732	Individual acceleration / deceleration of spindle synchronous for deceleration (S_{11} to Maximum speed)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}/s
[Valid data range] 0 to 100000

Set the acceleration/deceleration of spindle synchronous control with servo motor for deceleration of Leg 3 (S11 to Maximum speed). Acceleration switching speed 2 and acceleration switching speed 3 are the speeds set in parameter Nos. 25721 and 25722, respectively.

> **NOTE**
> When bit 6 (DCSx) of parameter No.11001 and bit 6 (CSA) of parameter No.11005 are set to 1, this parameter becomes effective.

	#7	#6	#5	#4	#3	#2	#1	#0
3704				SSS				

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off Before operation is continued.

#4 SSS Synchronous spindle control by each spindle is:
0: Not performed.
1: performed.
This master axis and slave axis of synchronous spindle control can be selected from the arbitrary spindles.
The target spindle of synchronous spindle control is specified in parameter No.4831.
In addition, the following signals affect the control.
Synchronous spindle signals of each spindle SPSYCs
Signals of synchronous control of the spindle phase for each spindle SPPHSs

<u>For this function, set 1.</u>

	#7	#6	#5	#4	#3	#2	#1	#0
4800			SCB					

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off Before operation is continued.

#5 SCB The combination of a master spindle and slave spindle for spindle synchronization depends on:
0: Setting of bit 4 (SSS) of parameter No.3704.
When bit 4 (SSS) of parameter No.3704 is set to 0
The first spindle and second spindle of each path can be selected as the master spindle and slave spindle, respectively, for spindle synchronization.
When bit 4 (SSS) of parameter No.3704 is set to 1 a combination of arbitrary spindles of each path can be selected for spindle synchronization.
Set a master spindle for each slave spindle in parameter No. 831.
Set a spindle number of each path. By setting a spindle number common to the system in parameter No.4832, an arbitrary spindle that belongs to a different path can be selected as a master spindle for spindle synchronization.
Set a spindle number common to the system. Set parameter No.4831 to 0.
Spindle synchronization based on arbitrary spindles must be enabled for the path to which a slave spindle belongs and for the path to which a master spindle belongs.

1: Conventional 16TT system compatible specifications.
The first spindle of path 1 and the first spindle of path 2 can be selected as the master spindle and slave spindle, respectively, for spindle synchronization.
As control signals, the signal interface of the 16TT system compatible specifications can be used.
For this function, set 0.

	#7	#6	#5	#4	#3	#2	#1	#0
4801								SNDs

[Input type] Parameter input
[Data type] Bit spindle

> **NOTE**
> When this parameter is set, the power must be turned off Before operation is continued.

#0 SNDs During spindle synchronization control, the rotation direction of each spindle motor is:
0: Same as the specified sign.
1: Opposite to the specified sign.

4810	Error pulse between two spindles when synchronizing phases in the spindle synchronization control mode

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 255

Set an allowable error pulse value between two spindles at phase synchronization time in the spindle synchronization control mode.
This parameter is used to check the completion of phase synchronization performed in the spindle synchronization control mode and to check the phase difference during spindle synchronization control. When the error pulse value between two spindles become equal to or less than the value set in this parameter, the spindle phase synchronization control completion signals FSPPH<Fn044.3> and FSPPH1 to 4 <Fn289.0 to 3> are set to “1”.

4811	Error pulse between two spindles when synchronizing phases in the spindle synchronization control mode

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 32767

Set the allowable error count for the error pulses between two spindles in the spindle synchronization control mode. This parameter is used to check a spindle synchronization error phase difference.
When a spindle synchronization error equal to or greater than the value set in this parameter is detected, the phase error monitor signals SYCAL<F044#4> and SYCAL1 to 4<F043#0 to 3> are set to 1.

4832	Master spindle of each slave spindle under spindle synchronization control (spindle number common to the system)

[Input type] Parameter input

[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled axes (common to the system)

> **NOTE**
> When this parameter is set, the power must be turned off Before operation is continued.

When a spindle is set as a slave spindle in spindle synchronization control on each spindle, set which spindle (master spindle) the slave spindle is to be synchronized with.

> **NOTE**
> 1 This parameter is valid if bit 4 (SSS) of parameter No. 3704 is set to 1 (to enable spindle synchronization based on arbitrary spindles) for the path to which a slave spindle belongs and for the path to which a master spindle belongs.
> 2 The setting of a slave spindle as a master spindle is invalid. Be sure to set 0 for a spindle that is to function as a master spindle.
> 3 In this parameter, set a spindle number common to the system. When this parameter is used, parameter No.4831 is set to 0.

Notes

> **NOTE**
> 1 Output signal SYCAL<Fn044.4> is used for monitoring a phase shift in synchronous control. Whether the processing is performed when a phase shift is detected depends on the specifications determined by the machine tool builder.
> 2 A gear ratio of 1 : 1 only is allowed between the spindle and position coder.
> 3 A CNC system restriction is imposed on the maximum number of spindles.
> 4 If the speed at the spindle synchronous mode exceeds maximum speed of slave spindle, an alarm PS0447 “ILLEGAL SETTING DATA” occurs.
> 5 Master spindle and slave spindle stop rotating due to a servo alarm, emergency stop, machine lock, etc.
> 6 When using this function, enable the spindle synchronous control function (bit 4 (SYC) of parameter No,8133 is 1).

10.20.10 Improvement of Spindle Stop Motion for Spindle Control with Servo Motor

Overview

If SV speed control mode is canceled during spindle rotation, spindle indexing is executed and spindle is stopped at specified position so far. When bit 0 (NSP) of parameter No.11002 is set to 1, If SV speed control mode is canceled during spindle rotation, spindle indexing is not executed and spindle is decelerated and stopped. When spindle indexing is unnecessary, machining cycle time can be reduced.

Format

When bit 0 (NSP) of parameter No.11002 is set to 1:

G96.1 P_ ; **The operation of the next block is started after deceleration stop of spindle.**

G96.2 P_ ; **The operation of the next block is started before deceleration stop of spindle.**

G96.1 P_ R_ ; **The operation of the next block is started after completing spindle indexing.**

G96.2 P_ R_ ; **The operation of the next block is started before completing spindle indexing.**

G96.3 P_ ; **The operation of the next block is started after confirming completion of spindle indexing.**

P : Spindle selection with multi-spindle control
R : Stoppage angle [deg] (0 to (parameter No.1260))

NOTE
1. When using G96.2, issue G96.3 before another spindle move command, to make sure that the spindle is at a complete halt.
2. To issue G96.1, G96.2, or G96.3, use an independent block.

Explanation

When bit 0 (NSP) of parameter No.11002 is set to 1, if SV speed is canceled during spindle rotation, spindle indexing is not executed and spindle is decelerated and stopped.

Command with a program

When R address is not commanded at G96.1 block, spindle indexing is not executed and spindle is decelerated and stopped. SV speed control mode in-progress signal SVREV<Fn521> becomes “0” after deceleration stop of spindle, and SV speed control mode is canceled. Then, the next block is started. When also R address is not commanded at G96.2 block as well, spindle indexing is not executed and spindle is decelerated and stopped. If G96.2 is commanded, the next block is started without waiting deceleration stop of spindle. If G96.3 is commanded, SV speed control mode in-progress signal SVREV<Fn521> becomes “0”, and SV speed control mode is canceled. When R address is commanded at G96.1 or G96.2 block, spindle indexing is executed and spindle is stopped at specified position.

Command with a signal

When SV speed control mode signal SRVON<Gn521> is set to “0”, spindle indexing is not executed and spindle is decelerated and stopped. SV speed control mode in-progress signal SVREV<Fn521> becomes “0” after deceleration stop of spindle, and SV speed control mode is canceled.

Acceleration up to deceleration stop of spindle

When SV speed control mode is canceled, parameter (No.11020, No.11021, No.11030, No.11031, No.11032) is applied to deceleration stop of spindle as well as acceleration of spindle.

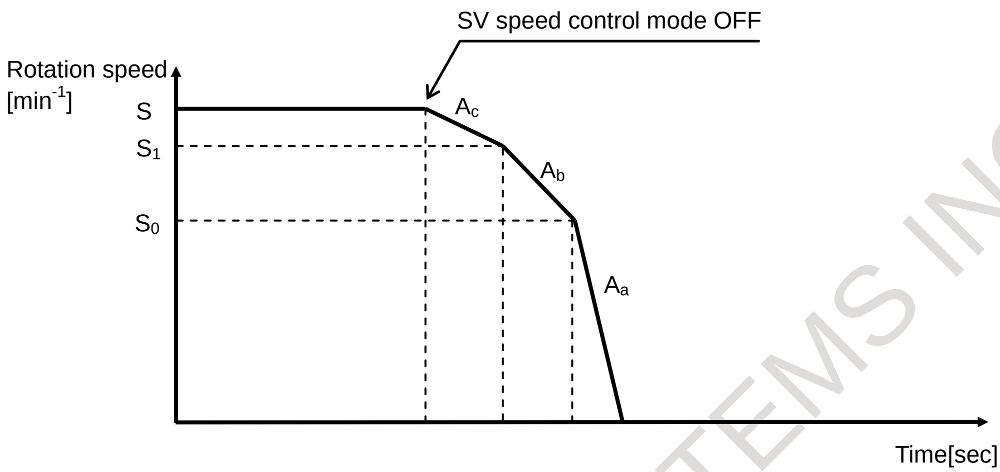


Fig. 10.20.10 Acceleration up to deceleration stop

Table 10.20.10 Parameters for acceleration up to deceleration stop

S	Specified rotation speed[min^{-1}]
S_0	Acceleration switching speed (S_0) [min^{-1}] (parameter No.11020)
S_1	Acceleration switching speed (S_1) [min^{-1}] (parameter No.11021)
A_a	Acceleration 1 [min^{-1}/s] (parameter No.11030)
A_b	Acceleration 2 [min^{-1}/s] (parameter No.11031)
A_c	Acceleration 3 [min^{-1}/s] (parameter No.11032)

Notes

- Specify an M code not involved in buffering as an M code to be used as a command for switching the SV speed control mode (parameters Nos.3411 to 3420 and 11290 to 11299).
- M code for switching the SV speed control mode must be specified in an independent block. When using multiple M commands in a single block, the M code must be specified as the first M command.
- G96.1, G96.2, G96.3, G96.4, and M code for switching the SV speed control mode must be specified in the path to which the spindle of interest belongs.
- After G96.2 command, command G96.3 before commanding move command such as rapid traverse (G00) or cutting feed (G01) to rotation axis. Move command is specified without commanding G96.3, alarm PS0445 “ILLEGAL AXIS OPERATION” is issued.
- If G96.3 is not commanded after G96.2 command, SV speed control mode is not canceled. For example, reset before G96.2 command.
- G96.1, G96.2 G96.3, G96.4 cannot be commanded for settings other than multi-spindle control type P.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11002								**NSP**

[Input type] Parameter input
[Data type] Bit axis

#0 NSP When SV speed control mode is canceled, spindle indexing is:
0: executed.
1: not executed.

10.20.11 Alarm and Message

Number	Message	Description
PS0003	TOO MANY DIGIT	Data entered with more digits than permitted in the NC instruction word. The number of permissible digits varies according to the function and the word.
PS0445	ILLEGAL AXIS OPERATION	The positioning command was issued in the speed control mode. Check the SV speed control mode signal.
PS0446	ILLEGAL COMMAND IN G96.1/G96.2/G96.3/G96.4	G96.1, G96.2, G96.3, and G96.4 are specified in the block that includes other commands. Modify the program.
PS0447	ILLEGAL SETTING DATA	The spindle controlled axis is incorrectly set. Check the parameter for the live tool control function by a servo motor.
PS0601	ILLEGAL AXIS COMMAND FOR SERVO MOTOR SPINDLE	The move command was executed to the servo axes for spindle use of the spindle control with servo motor. Modify the program.
SV0410	EXCESS ERROR (STOP)	The amount of positional deviation during stopping exceeded the parameter No. 1829 setting value.
SV0411	EXCESS ERROR (MOVING)	The amount of positional deviation during traveling became excessive than the parameter setting value.
SV0417	ILL DGTL SERVO PARAMETER	A digital serve parameter setting is incorrect.
DS2003	ILLEGAL USE FOR SERVO MOTOR SPINDLE	The servo axes for spindle use of the spindle control with servo motor was used by the following functions. - PMC axis control - Chopping
PW0036	ILLEGAL SETTING FOR SERVO MOTOR SPINDLE	The parameter setting for the servo axes for spindle use of the spindle control with servo motor is illegal. - The servo axes number for spindle use has exceeded 4 axes. - Servo axes number for spindle use are fewer than the axes number of designation of servo axes for spindle use. - The servo axes for spindle use is not set as a spindle control with servo motor axis. - The servo axes for spindle use is not set as a rotation axis.. - The servo axis number (parameter No.1023) is a negative value.

10.21 SPINDLE REVOLUTION NUMBER HISTORY FUNCTION

Overview

This function counts the spindle revolution number and displays the total revolution number as diagnosis data.
Total revolution number data as diagnosis data can be read with the PMC window function, and can be used for spindle unit life management on the PMC ladder.

Explanation

This function is enabled by setting bit 5 (SSH) of parameter No. 3799, to 1.

Two total revolution number displays are provided, which are shown as diagnosis data Nos. 1520 and 1521, respectively.

Total revolution number data as diagnosis data can be read with the PMC window function.

For details, refer to "Reading diagnosis data (function code 33) in Chapter 5, "Window Functions", in the PMC Programming Manual (B-64513EN).

As for total revolution number display data, the total revolution number data for a specified diagnosis data number can be cleared for a specific spindle, using the signals below.

- Using the total spindle revolution number reset selection signal SSRS <Gn533.4>, select the diagnosis data number of the total revolution number data to be cleared.
- Using an appropriate total spindle revolution number reset signals SSR1 to SSR4 <Gn533.0 to Gn533.3>, clear the total revolution number data for a specific spindle.

NOTE

1. When using this function, enable the spindle serial output (bit 5 (SSN) of parameter No,8133 is 0).
2. Storage of spindle revolution number data in data-retention memory is performed at intervals of about one second. If the power is turned off while a spindle is revolving, revolution number data is not stored for about one second before the power is turned off.
3. This function is effective to a built-in spindle motor and to a spindle configuration in which a spindle and a spindle motor is connected together at a gear ratio of 1:1.
4. If the spindle switching function (a configuration in which one set of main/sub-spindle motors are connected to a single spindle amplifier) is used, total revolution number data is counted as an integrated value of two spindles.

Signal

Total spindle revolution number reset signals SSR1 to SSR4 <Gn533.0 to Gn533.3>

[Classification] Input signal

[Function] Each of these signals resets the total revolution number for its corresponding spindle, returning diagnosis data No. 1520, total revolution number display, to zero.
If SSRS is “1”, each of these signals resets diagnosis data No. 1521, total revolution number display, returning it to zero.

[Operation] When set to “1”, each signal performs the operation below.

- Reset the total revolution number for its corresponding spindle. SSR1 corresponds to the first spindle, SSR2 the second spindle, SSR3 the third, and SSR4 the fourth.

Total spindle revolution number reset selection signal SSRS <Gn533.4>

[Classification] Input signal

[Function] Used to select between diagnosis data Nos. 1520 and 1521; the total spindle revolution number for the selected item No. will be reset.

[Operation] When set to “1”, it performs the operation below.

- The total spindle revolution number for diagnosis data No. 1521 can be reset.

When set to “0”, it performs the operation below.

- The total spindle revolution number for diagnosis data No. 1520 can be reset.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn533				SSRS	SSR4	SSR3	SSR2	SSR1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3799			SSH					

[Input type] Parameter input
[Data type] Bit spindle

#5 SSH On the diagnosis screen, total spindle speed data is:
0: Not displayed.
1: Displayed.

Diagnosis data

1520	Total spindle revolution number 1

[Data type] 2-word spindle
[Unit of data] 1000 min^{-1}
[Valid data range] 0 to 999999999
The spindle revolution number is counted, and the total revolution number is displayed.

1521	Total spindle revolution number 2

[Data type] 2-word spindle
[Unit of data] 1000 min^{-1}
[Valid data range] 0 to 999999999
The spindle revolution number is counted, and the total revolution number is displayed.

10.22 SERVO/SPINDLE SYNCHRONOUS CONTROL

10.22.1 Servo/Spindle Synchronous Control

Overview

This function provides the following functions to use a servo motor as a spindle:

(a) Servo motor spindle control
Can rotate the servo motor at the rotation speed specified with an input signal.

(b) Servo motor spindle synchronization
Can rotate the servo motor in synchronization with the feedback pulses from the position coder of the spindle.

(c) Differential speed synchronization
(i) Can superimpose a command from the CNC on the servo motor in servo motor spindle synchronization.
(ii) Can superimpose the rotation speed specified with an input signal on the servo motor in servo motor spindle synchronization.

NOTE
When using this function, enable the spindle synchronous control (bit 4 (SYC) of parameter No,8133 is 1).

Explanation

- Connection

Input feedback pulses from the position coder to the pulse module.
To make effective the feedback pulses input to the pulse module, set parameters Nos. 24096 to 24103.
This causes the spindle (position coder) and the servo motor to synchronize with each other.

Example) Connection example

Parameter No. 24096 = 1: Uses connector number 1 (JF101).

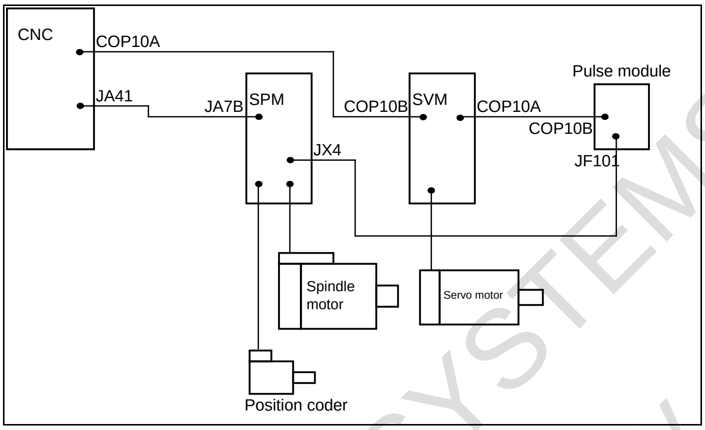


Fig. 10.22.1 (a) Connection example for servo/spindle synchronous control

> **NOTE**
> During operation, the combination of spindle and servo motor to synchronize cannot be changed.

- Configuration

To use this function, the detection unit must be specified by setting the flexible feed gear (M/N) (parameter No.2084 and 2085) in such a way that the number of pulses per rotation of the spindle will be 4096. Thus, it is necessary to select a servo motor-to-spindle gear ratio a:b so that the M and N values of the flexible feed gear as determined from the expression below will be equal to or less than 32767.

$$\frac{M}{N} = \frac{a}{b} \times \frac{4,096}{1,000,000}$$

To perform servo motor and spindle synchronous control, set a magnification (integral multiple or R) so that the number of feedback pulses from the position coder will be 4096 per rotation of the spindle. That is, the gear ratio of the position coder to the spindle must be one to an integer.

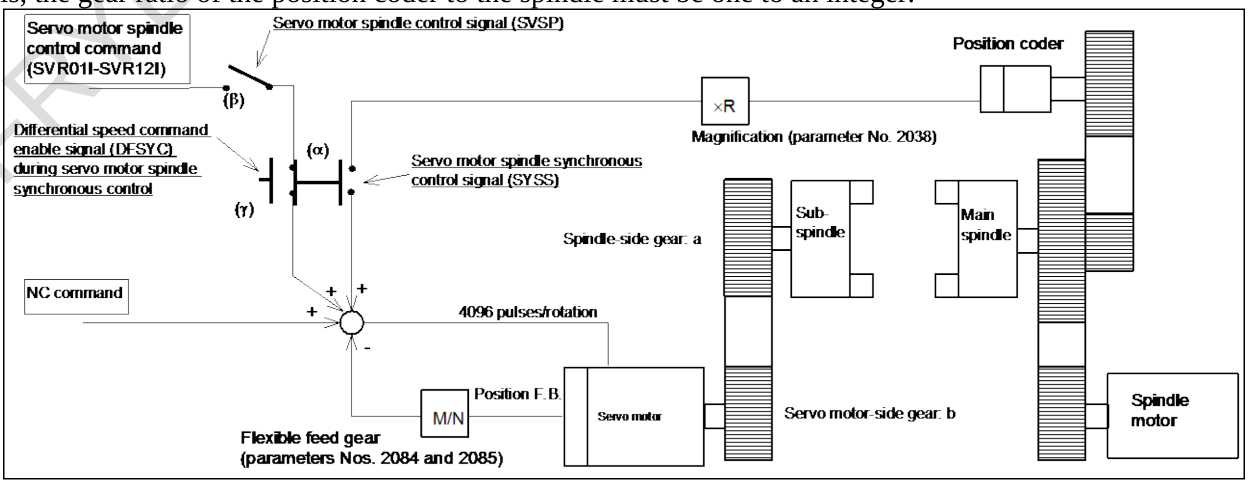


Fig. 10.22.1 (b) System configuration example

- Setting example

If using the A-axis (rotation axis) of the servo motor as the sub-spindle

Conditions

Detector	:	1,000,000 (pulses/rotation)
Spindle-side gear	:	8,000
Servo motor-side gear	:	3,600
CMR	:	1 (setting: 2)

Gear ratio of the position coder to the main-spindle: 1 to 2

The flexible feed gear can be determined from the formula below.

$$\frac{M}{N}=\frac{a}{b}\times\frac{4{,}096}{1{,}000{,}000}=\frac{8{,}000}{3{,}600}\times\frac{4{,}096}{1{,}000{,}000}=\frac{256}{28{,}125}$$

If these settings are made, a normal move command for the A-axis will be A4.096 per rotation of the spindle, and the speed command will be a feed per minute command per rpm, F4.096.

The magnification (R) for servo motor and spindle synchronous control will be set to 2 because the gear ratio of the position coder to the main-spindle is 1:2.

- Servo motor spindle control

Can rotate the servo motor at the rotation speed specified with an input signal.

- Servo motor spindle control mode

By setting the servo motor spindle switching signal SVSP <Gn022.7> to "1", the system enters servo motor spindle control mode, so that the servo motor can be rotated at the rotation speed specified with a signal.
In servo motor spindle control mode, the servo motor spindle control mode signal SVSPM <Fn090.7> is set to "1".
By setting SVSP to "0", servo motor spindle control mode is canceled. If this occurs, SVSPM is set to "0".

NOTE

1. In servo motor spindle control mode, coordinates will not be updated. Thus, the relationships between coordinates and actual machine position will be lost.
2. If a move command is commanded from the CNC on the servo motor in servo motor spindle control mode, alarm PS5211 "ILLEGAL AXIS OPERATION" is issued.

- Specifying a motor rotation speed

The rotation speed of the servo motor in servo motor spindle control mode can be specified with the servo motor rotation speed specification signals SVR01I to SVR12I <Gn021.0 to Gn022.3>. Set the setting determined from the formula below, in binary notation.

$$Setting=\frac{\text{Speed of spindle motor}}{\text{Maximum speed of spindle motor(parameter No.3842)}}\times 4095$$

The rotation direction of the motor can be specified with the servo motor rotation polarity specification signal SVGN <Gn022.5>.

- Acceleration/deceleration

If servo motor spindle control mode is turned on or off or if the rotation speed specified with SVR01I to SVR12I changes, acceleration/deceleration is performed with the time constant set in parameter No. 3843.
When a steady state is attained after the completion of acceleration/deceleration, the servo motor spindle control mode acceleration/deceleration completion signal SVAR <Fn090.6> is set to "1".

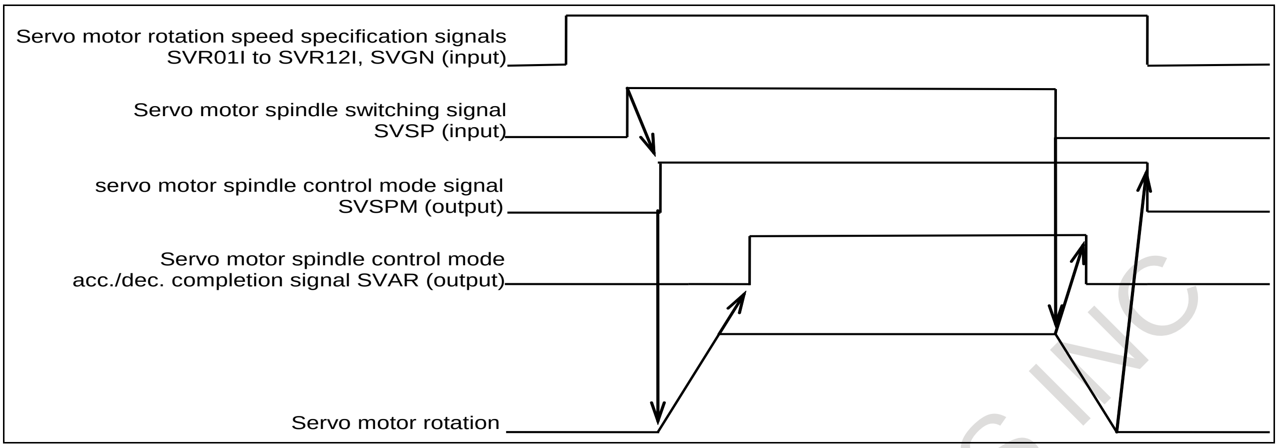


Fig. 10.22.1 (c) Timing chart of signals during servo motor spindle control

- Servo motor spindle synchronization

Can rotate the servo motor in synchronization with the feedback pulses from the position coder of the main-spindle.

- Servo motor spindle synchronization mode

By setting the servo motor spindle synchronization start signal SYSS <Gn061.2> to “1”, the system enters servo motor spindle synchronization mode. When the system enters servo motor spindle synchronization mode, the servo motor accelerates or decelerates to the rotation speed of the main-spindle that is specified with parameter No. 3844. If a move command is commanded from the CNC in acceleration/deceleration, alarm PS5211 “ILLEGAL AXIS OPERATION” is issued. Upon completion of acceleration/deceleration, the servo motor spindle synchronization mode acceleration/deceleration completion signal SYAR <Fn090.4> is set to “1”. Then the servo motor rotates in synchronization with the feedback pulses from the position coder.

When the system enters servo motor spindle synchronization mode, the servo motor spindle synchronization mode signal SYSSM <Fn090.5> is set to “1”.

By setting SYSS to “0”, servo motor spindle synchronization mode is canceled. If a rotation command is being executed with servo motor spindle control, the sub-spindle accelerates or decelerates to the speed specified at that point. Upon completion of the cancellation of servo motor spindle synchronization, SYSSM is set to “0”.

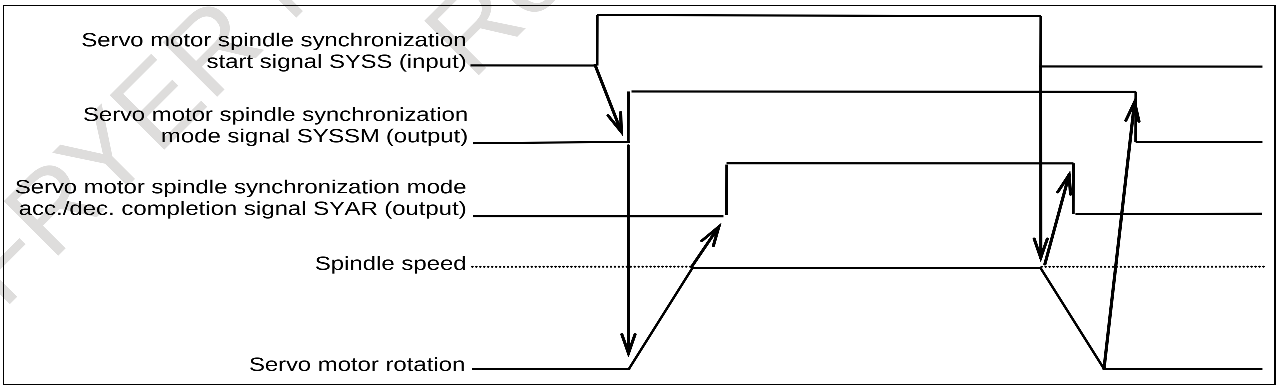


Fig. 10.22.1 (d) Timing chart of signals during servo motor spindle synchronization

> **NOTE**
> 1 In servo motor spindle synchronization mode, coordinates will not be updated. Thus, the relationships between coordinates and actual machine position will be lost. That is, the updating of coordinates with NC commands in servo motor spindle synchronization mode does not represent the actual movements of the machine.

> **NOTE**
> 2 At the start or end of servo motor spindle synchronization mode, all servo axes except sub-spindles must be stopped.
> 3 To start or cancel servo motor spindle synchronization mode during automatic operation, be sure to specify an M code that does not perform buffering before and after switching modes.
> 4 In servo motor spindle synchronization mode, nano interpolation is disabled on sub-spindles.
> 5 In servo motor spindle synchronization mode, do not use the torque control or speed control of PMC axis control on sub-spindles.

- Relationships with servo motor spindle control mode

In servo motor spindle synchronization mode, rotation commands due to servo motor spindle control commands are disabled. By setting SYSS to 0, servo motor spindle control commands are enabled.

- Differential speed synchronization

Can superimpose a move command from the CNC or the rotation speed specified with an input signal on the servo motor in servo motor spindle synchronization mode. It cannot superimpose both of them at the same time.

- Superimposition of a move command

If a move command is executed on the servo axis in servo motor spindle synchronization mode, the move command is superimposed on the synchronization speed.

- Superimposition of the rotation speed from an input signal

By setting the differential speed synchronization command signal DFSYC <Gn022.4> and the servo motor spindle switching signal SVSP to “1” in servo motor spindle synchronization mode, SVR01I to SVR12I and SVGN become effective, so that the servo motor rotates at the speed resulting from synthesizing the speed specified with SVR01I to SVR12I and the speed at which to synchronize with the feedback from the position coder.

If a move command is commanded from the CNC on the servo motor in servo motor spindle synchronization mode when DFSYC and SVSP signals set to “1”, alarm PS5211 “ILLEGAL AXIS OPERATION” is issued.

- Relationships with Servo motor spindle control and Servo motor synchronous control

In servo motor spindle synchronization mode, rotation commands due to servo motor spindle control commands are disabled. (When servo motor synchronous control is performed during servo motor spindle control mode, servo motor spindle control mode is canceled).

By one of the followings, servo motor spindle control commands are enabled again.

- Differential speed synchronization mode is enabled (Change DFSYC from "0" to "1")
- Servo motor synchronous control is canceled (Change SYSS from "1" to "0")

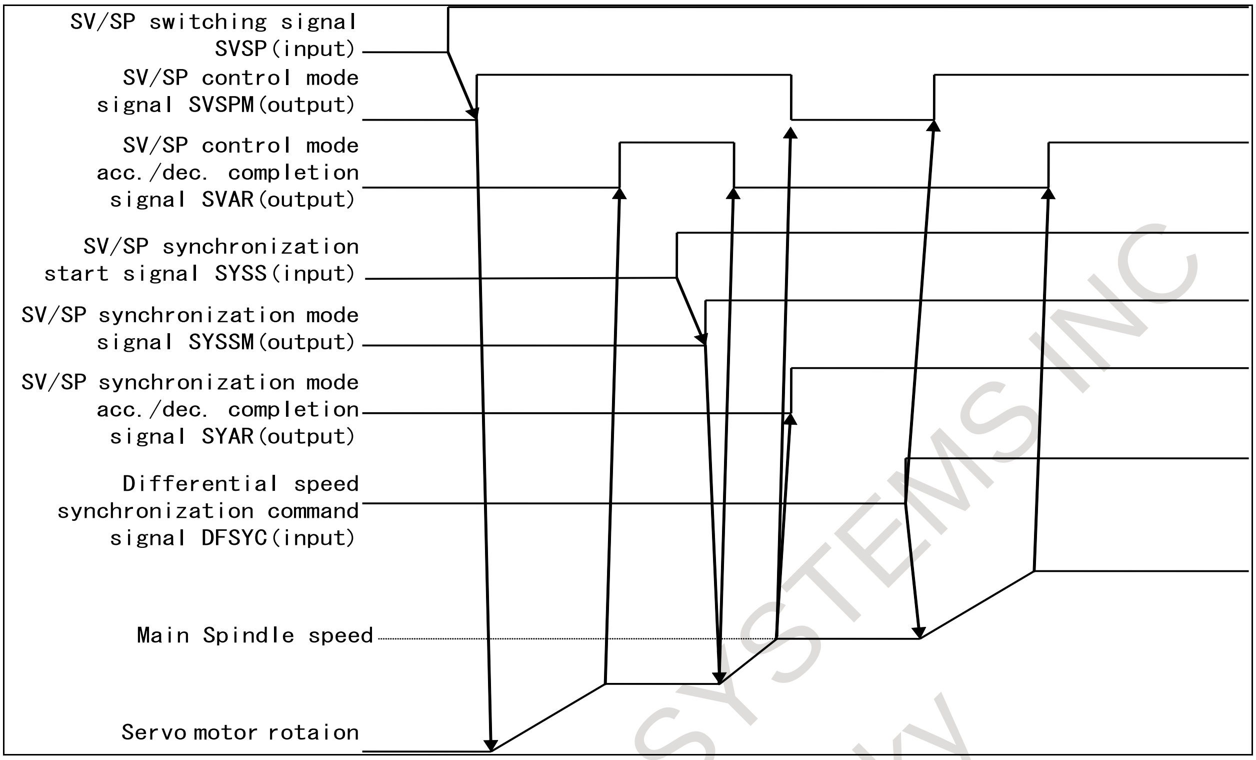


Fig.10.22.1 (e) Timing chart of signals related to servo motor spindle control

Feed forward

During servo spindle control and servo spindle synchronization, the function of feed forward with a servo motor can be enabled. Enabling the feed forward function has the effect of preventing delays due to servo errors when differential speed commands are issued.

Signal

Servo motor spindle control switching signal SVSP <Gn022.7>

[Classification] Input signal

[Function] By setting SVSP to "1", the system enters servo motor spindle control mode, so that a command is executed on the servo motor at the rotation speed previously set in the 12 bits of SVR1I to SVR12I. If this occurs, SVSPM is set to "1".
By setting SVSP to "0", servo motor spindle control mode is canceled. If this occurs, SVSPM is set to "0".

> **NOTE**
> In servo motor spindle control mode, coordinates will not be updated. Thus, the relationships between coordinates and actual machine position will be lost.

Servo motor spindle control mode signal SVSPM <Fn090.7>

[Classification] Output signal

[Function] This signal notifies that the system is in servo motor spindle control mode.

[Output cond.] This signal is set to "1" when the system enters servo motor spindle control mode.

Servo motor rotation speed specification signals SVR01I to SVR12I <Gn021.0 to Gn022.3>

[Classification] Input signal

[Function] These signals enable specification of the rotation speed of the servo motor during servo motor spindle control by setting the setting determined from the formula below, in binary notation.

$$Setting = \frac{\text{Speed of spindle motor}}{\text{Maximum speed of spindle motor(parameter No.3842)}} \times 4095$$

Servo motor rotation polarity specification signal SVGN <Gn022.5>

[Classification] Input signal

[Function] This signal specifies the rotation direction of the servo motor.

0: The polarity is plus.

1: The polarity is minus.

Servo motor spindle control mode acceleration/deceleration completion signal SVAR <Fn090.6>

[Classification] Output signal

[Function] This signal notifies that servo motor acceleration/deceleration has ended in servo motor spindle control.

[Output cond.] This signal is set to “1” at the end of acceleration/deceleration in servo motor spindle control mode.

Servo motor spindle synchronization start signal SYSS <Gn061.2>

[Classification] Input signal

[Function] By setting SYSS to “1”, the system enters servo motor spindle synchronization mode. If this occurs, SYSSM is set to “1”.

When the system enters servo motor spindle synchronization mode, the sub-spindle accelerates or decelerates to the rotation speed of the main spindle. Then, the sub-spindle rotates in synchronization with the feedback pulses from the position coder of the main spindle.

By setting SYSS to “0”, servo motor spindle synchronization mode is canceled. If a rotation command is being executed with servo motor spindle control, the sub-spindle accelerates or decelerates to the speed specified at that point. Upon completion of the cancellation of servo motor spindle synchronization, SYSSM is set to “0”.

In servo motor spindle synchronization mode, rotation commands due to servo motor spindle control commands are disabled. By setting SYSS to “0”, servo motor spindle control commands are enabled.

> **NOTE**
> In servo motor spindle synchronization mode, coordinates will not be updated. Thus, the relationships between coordinates and actual machine position will be lost. That is, the updating of coordinates with NC commands in servo motor spindle synchronization mode does not represent the actual movements of the machine.

Servo motor spindle synchronization mode signal SYSSM <Fn090.5>

[Classification] Output signal

[Function] This signal notifies that the system is in servo motor spindle synchronization mode.

[Output cond.] This signal is set to “1” when the system enters servo motor spindle synchronization mode.

Servo motor spindle synchronization mode acceleration/deceleration completion signal SYAR <Fn090.4>

[Classification] Output signal

[Function] This signal notifies that servo motor acceleration/deceleration has ended in servo motor spindle synchronization.

[Output cond.] This signal is set to “1” at the end of acceleration/deceleration in servo motor spindle synchronization mode.

Differential speed synchronization command signal DFSYC <Gn022.4>

[Classification] Input signal

[Function] This signal enables superimposing of rotation commands due to servo motor spindle control in servo motor spindle synchronization mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn021	SVR08I$^{\#P}$	SVR07I$^{\#P}$	SVR06I$^{\#P}$	SVR05I$^{\#P}$	SVR04I$^{\#P}$	SVR03I$^{\#P}$	SVR02I$^{\#P}$	SVR01I$^{\#P}$
Gn022	SVSP$^{\#P}$		SVGN$^{\#P}$	DFSYC$^{\#P}$	SVR12I$^{\#P}$	SVR11I$^{\#P}$	SVR10I$^{\#P}$	SVR09I$^{\#P}$
Gn061						SYSS$^{\#P}$		

	#7	#6	#5	#4	#3	#2	#1	#0
Fn090	SVSPM$^{\#P}$	SVAR$^{\#P}$	SYSSM$^{\#P}$	SYAR$^{\#P}$				

Parameter

3841	Servo motor spindle control axis number

[Data type] Word

[Valid data range] 1 to 24

This parameter sets the axis number of an axis to be subject to servo motor spindle control or servo motor spindle synchronization.
Setting the parameter to 0 disables servo motor spindle control and servo motor spindle synchronization.
For servo motor spindle synchronization, you have to set bit 4 (SPSx) of parameter No. 2016.

3842	Maximum speed for servo motor spindle control

[Data type] 2-word

[Unit of data] min^{-1}

[Valid data range] 0 to 9999

This parameter sets the maximum speed of the spindle to be subject to servo motor spindle control.

3843	Time constant for acceleration/deceleration under servo motor spindle control

[Data type] Word

[Unit of data] msec

[Valid data range] 0 to 4000

This parameter sets the time constant for acceleration/deceleration under servo motor spindle control and servo motor spindle synchronization.
The type of acceleration/deceleration is linear acceleration/deceleration.
Set the parameter to the time to be taken for the spindle speed to reach 1000 (min^{-1}).

3844	Master spindle number

[Data type] Word

[Valid data range] 0 to 104

Set the number of the spindle (position coder) to be subject to servo motor spindle synchronization.
The hundreds and tens digits represent a path; the units digit represents the number of the position coder in the path.
To synchronize the servo axis in the second path and the second position coder in the first path under dual-path control, for example, set this parameter for the second path to 12.
When the hundreds and tens digits are both 0, the local path is assumed.
When a value of 0 is specified, the first position coder in the local path is assumed.

> ⚠ **WARNING**
> 1 The combination of the position coder and servo motor to be synchronized with each other is determined by wire connection. Even though you change the setting of this parameter, therefore, you cannot change the combination of the position coder and servo motor to be synchronized with each other.
> 2 This parameter is used for acceleration or deceleration to be performed when the synchronization mode is turned on/off.
> 3 It is dangerous to set a value not matching the actually wire-connected combination as it prevents correct acceleration/deceleration. Be sure to set a value matching the actual wire connection.

	#7	#6	#5	#4	#3	#2	#1	#0
2003					PIENx			

[Data type] Bit axis

#3 PIENx PI control is:
0: Disabled.
1: Enabled.

> **NOTE**
> This bit must be set to use the feed forward function.

	#7	#6	#5	#4	#3	#2	#1	#0
2005							FEEDx	

[Data type] Bit axis

#1 FEEDx Feed forward function is:
0: Disabled.
1: Enabled.

> **NOTE**
> 1 This bit must be set to use the feed forward function.
> 2 This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

	#7	#6	#5	#4	#3	#2	#1	#0
2011							FFALx	

[Data type] Bit axis

#1 FFALx Feed-forward control is:

0: Enabled only in normal cutting feed during automatic operation when parameter No.1800#3 is set to 0.
Enabled in normal cutting feed and rapid traverse during automatic operation when parameter No.1800#3 is set to 1.

1: Always enabled regardless of the mode.

> **NOTE**
> This bit must be set to use the feed forward function on Servo/ spindle synchronization.

	#7	#6	#5	#4	#3	#2	#1	#0
2016				SPSx				

[Data type] Bit axis

#4 SPSx Servo motor spindle synchronization is:

0: Disabled.

1: Enabled.

> **NOTE**
> Be sure to set this bit to perform servo spindle synchronization.

2038	Spindle feedback magnification (R)

[Data type] Word
[Valid data range] ±32737

Set a magnification so that the number of feedback pulses from the position coder of the main spindle during servo motor spindle synchronization will be 4096 per rotation of the spindle.

$$\text{Setting} = \frac{\text{Gear of spindle}}{\text{Gear of position coder}}$$

If the setting is 0, synchronous control is not performed.
The rotation direction of the servo motor changes with the sign of the setting.

2068	Feed forward coefficient

[Data type] Word
[Valid data range] 0 to 10000

Setting = α x 100

> **NOTE**
> To use the feed forward function, this parameter must be set.

2069	Velocity feed forward coefficient

[Data type] Word
[Valid data range] 50 to 200

This parameter adjusts follow-up capability if it is poor.

NOTE
To use the feed forward function, this parameter must be set.

2084	Flexible feed gear (numerator) (M)
2085	Flexible feed gear (denominator) (N)

[Data type] Word axis
[Valid data range] 0 to 32767
Set the flexible feed gear so that the command per rotation of the spindle will be 4096.

24096	Connector number for the first or ninth separate detector interface unit
24097	Connector number for the second or tenth separate detector interface unit
24098	Connector number for the third or eleventh separate detector interface unit
24099	Connector number for the fourth or twelfth separate detector interface unit
24100	Connector number for the fifth separate detector interface unit
24101	Connector number for the sixth separate detector interface unit
24102	Connector number for the seventh separate detector interface unit
24103	Connector number for the eighth separate detector interface unit

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 8
Set a connector number for the connector to which a separate detector interface unit is attached if the separate detector interface unit is to be used. The following table lists the necessary settings. Be sure to specify 0 for connectors not in use.

Correspondence between connectors and connector numbers

Connector	Connector number
JF101	1
JF102	2
JF103	3
JF104	4
JF105	5
JF106	6
JF107	7
JF108	8

(Setting example)

Controlled axis	Connector to which each separate detector interface unit is attached				Parameter setting			
	1st connector	2nd connector	5th connector	6th connector	No. 24096	No. 24097	No. 24100	No. 24101
X1	JF101	–	–	–	1	0	0	0
Y1	–	JF102	–	–	0	2	0	0
Z1	–	–	JF102	–	0	0	2	0

Controlled axis	Connector to which each separate detector interface unit is attached				Parameter setting			
	1st connector	2nd connector	5th connector	6th connector	No. 24096	No. 24097	No. 24100	No. 24101
X2	–	JF101	–	–	0	1	0	0
Y2	–	–	–	JF101	0	0	0	1
Z2	–	–	–	–	0	0	0	0
A1	–	–	JF101	–	0	0	1	0
B1	–	–	–	JF102	0	0	0	2
C1	–	JF104	–	–	0	4	0	0
A2	JF102	–	–	–	2	0	0	0
B2	–	JF103	–	–	0	3	0	0
C2	–	–	–	JF103	0	0	0	3

NOTE

1 Specify these parameters when separate detector interface units are used.

2 Parameters Nos. 24096 to 24103 are specified automatically when data is entered on the FSSB setting screen if the FSSB setting mode in use is the automatic setting mode (bit 0 (FMD) of parameter No. 1902 = 0). If the manual setting 2 mode (bit 0 (FMD) of parameter No. 1902) = 1), specify the parameters directly.

Alarm and message

Number	Message	Description
PS5211	ILLEGAL AXIS OPERATION	- A move command is commanded from the CNC on the servo motor in servo motor spindle control mode. - A move command is commanded from the CNC on the servo motor in acceleration/deceleration of servo motor spindle synchronization mode. - A move command is commanded from the CNC on the servo motor in servo motor spindle synchronization mode when differential speed synchronization command signal DFSYC and the servo motor spindle switching signal SVSP set to “1”.

Notes

Servo/Spindle synchronous control (FSSB type)

The separate detector interface unit becomes unnecessary by the FSSB communication between a CNC and a spindle amplifier.

However, if the separate detector interface unit is used, do not set the setting (the parameters FHRSV (bit 2 of No.2429) and FHESP (bit 2 of No.4549) of Servo/Spindle synchronous control (FSSB type).

Refer to "11.22.2 Servo/Spindle synchronous control (FSSB type)" for details.

Servo alarm

If a servo alarm occurs, servo spindle control and servo spindle synchronization mode are canceled. If this occurs, SVSPM, SYSSM, SVAR, and SYAR (<Fn090> signals) are all cleared.

After the servo alarm is canceled, servo spindle control and servo spindle synchronization will be enabled again.

Servo off

If follow-up is performed (bit 0 (FUPx) of parameter No. 1819 = 0), servo spindle control and servo spindle synchronization mode are canceled. If this occurs, SVSPM, SYSSM, SVAR, and SYAR (<Fn090> signals) are all cleared.

If follow-up is not performed (bit 0 (FUPx) of parameter No. 1819 = 1), servo spindle control and servo spindle synchronization mode are not canceled. Commands are accumulated as servo errors.

Emergency stop

In the event of an emergency stop, servo spindle control and servo spindle synchronization mode are canceled. If this occurs, SVSPM, SYSSM, SVAR, and SYAR (<Fn090> signals) are all cleared.
After the emergency stop is canceled, servo spindle control and servo spindle synchronization will be enabled again.

Multi-path

- Number of axes

Only a single servo axis per path can be used with this function.

- Signal address

If the servo motor on the second path is used, the addresses of all control signals are the original ones plus 1000.
Similarly, for the third path, the addresses are the original ones plus 2000.

- Synchronization between paths

It is possible to synchronize between paths. To do this, input the signal for the path to which the servo axis belongs.
For example, to synchronize the spindle on path 1 with the servo axis on path 2, input the signal on path 2 (G1000 to G1999).

- Synchronization of multiple axes

It is possible to synchronize multiple servo axes to a single main-spindle. To do this, input the signals on the multiple paths to which the servo axes belong.
It is necessary to connect multiple position coders to the same main-spindle and input them to the connectors of the respective pulse modules.

10.22.2 Servo/Spindle Synchronous Control (FSSB Type)

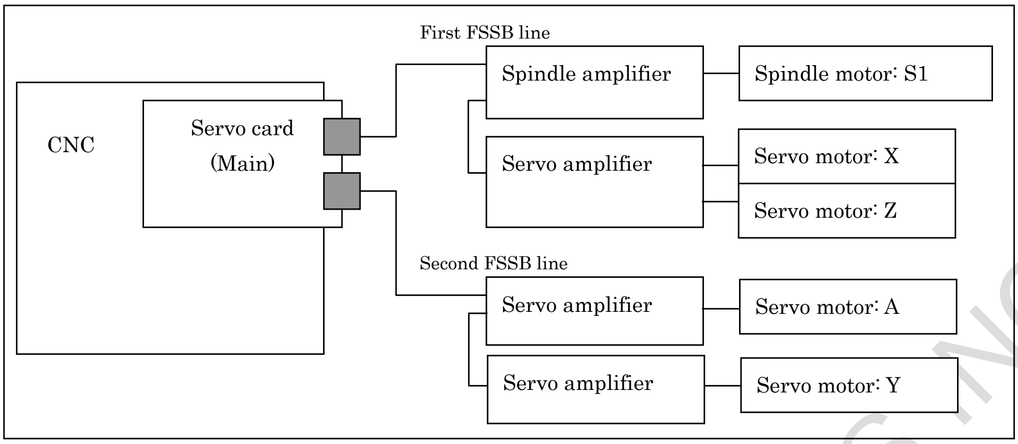
Outline

By the FSSB communication between a CNC and a spindle amplifier, rotational position information of a spindle can be transmitted from a spindle amplifier to a servo control on CNC. Servo/Spindle synchronous control (FSSB type) is a function that a servo axis follows a spindle axis by utilizing this FSSB communication. And it is a feature of this function that the separate detector interface unit becomes unnecessary compared with the ordinary function.

Explanation

The amplifier of main-spindle (spindle motor) and the amplifier of sub-spindle (servo motor) used for Servo/Spindle synchronous control (FSSB type) need to be connected to the main servo card. The spindle amplifier and the servo amplifier used for Servo/Spindle synchronous control (FSSB type) can be connected to both the first FSSB line and the second FSSB line. (This function can be executed between two motors connected to another line.)

Ex.1) Servo/Spindle synchronous control (FSSB type) with A axis and S1 spindle axis in the one path system.
[Example of connection of FSSB]

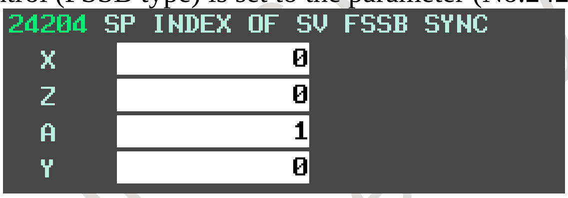


[Example of parameter setting]

The following setting are necessary in addition to setting of existing parameters related to Servo/Spindle synchronous control.

When the parameter FHR (Bit 0 of No.24203) is set to 1, position data transmission by FSSB is enabled.

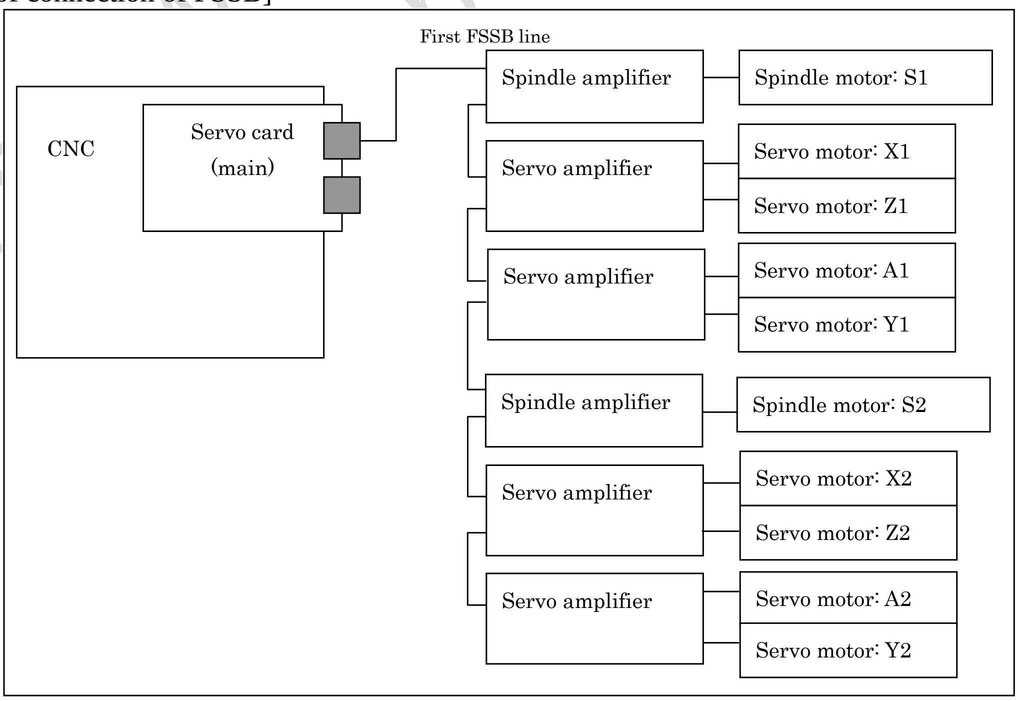
And, the index number of the spindle axis (S1) that synchronizes with the servo axis (A) used by Servo/Spindle synchronous control (FSSB type) is set to the parameter (No.24204) as follows.



In addition, set the parameter FSBSYN (bit2 of No.2429) of A axis and the parameter FHESP (bit2 of No.4549) of S1 spindle axis to 1.

Ex.2) Servo/Spindle synchronous control (FSSB type) with A1 - S1 and A2 - S2 in two path system.

[Example of connection of FSSB]

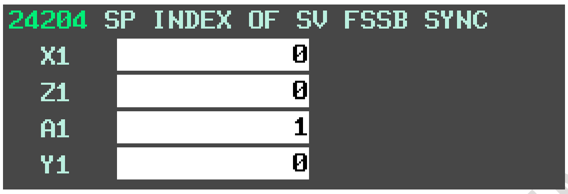


[Example of parameter setting]

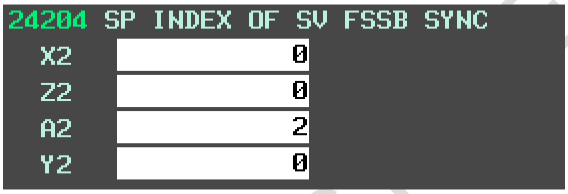
The following setting are necessary in addition to setting of existing parameters related to Servo/Spindle synchronous control.

When the parameter FHR (Bit 0 of No.24203) is set to 1, position data transmission by FSSB is enabled.
And, the index number of the spindle axes (S1, S2) that synchronizes with the servo axes (A1, A2) used by Servo/Spindle synchronous control (FSSB type) is set to the parameter (No.24204) as follows.

Path1



Path2



In addition, set the parameter FSBSYN (bit2 of No.2429) of A1 and A2 axes and the parameter FHESP (bit2 of No.4549) of S1 and S2 spindle axes to 1.

Eight or less index numbers of the spindle axis can be set in all system. If nine or more index numbers of the spindle axis or illegal index number of spindle axis is set, the alarm (PW0037) "SV/SP COMBINATION ERROR" is issued.

NOTE

1. When the parameter is changed, the power must be turned off before an operation is continued.
2. Servo/Spindle synchronous control (FSSB type) cannot be used with analog spindle or the spindle control with servo motor. If the above setting is set the parameter No.24204, the alarm (PW0037) “SV/SP COMBINATION ERROR” is issued.
3. The setting of the arbitrary gear ratio (DMR function) for main-spindle (spindle motor) cannot be used.
4. With sub-spindle (servo motor) used by Servo/Spindle synchronous control (FSSB type), the following servo software functions cannot be used.
   - FSSB high-speed rigid tapping
   - Electronic Gear Box
   - Full-closed control

   If these functions are used, the alarm SV0417, “ILL DGTL SERVO PARAMETER” is issued.
5. Others follow notes of Servo/Spindle synchronous control.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
24203								FHR

> **NOTE**
> When the parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 FHR Position data transmission by FSSB is:
0: Disabled.
1: Enabled.

> **NOTE**
> 1 In addition, it is necessary to set parameter No.24204.
> 2 When Servo/Spindle synchronous control (FSSB type) is used, set this parameter to 1.

24204	The index number of the spindle axis that synchronizes to each servo axis

> **NOTE**
> When the parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to the maximum number of spindles

Set the index number of the spindle motor (main-spindle) that synchronizes with the servo axis (sub-spindle) by Servo/Spindle synchronous control (FSSB type).

> **NOTE**
> 1 When the parameter FHR (Bit 0 of No.24203) is 1, this parameter is enabled.
> 2 Eight or less index numbers of the spindle axis can be set in the system. If nine or more index numbers of the spindle axis are set, alarm (PW0037) “SV/SP COMBINATION ERROR” is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
2429						FSBSYN		

[Input type] Parameter input
[Data type] Bit axis

#2 FSBSYN Servo control by Servo/Spindle synchronous control (FSSB type) is:
0: Disabled.
1: Enabled.

> **NOTE**
> 1 When using Servo/Spindle synchronous control (FSSB type), set this parameter to 1.
> 2 In the servo axis, in which this parameter is 1, the following function cannot be used. If using with either of the following functions, the alarm (SV0417) “ILL DGTL SERVO PARAMETER” is issued. (Detail number of diagnosis data No.352 becomes 4292 or 4291)
>   - FSSB high-speed rigid tapping (Bit 1 of No.2429)
>   - Electronic gear box (Bit 0 of No.2011)
>   - Full-closed control (Bit 1 of No.1815)
>
> 3 When the parameter is set, the power must be turned off before operation is continued.

	#7	#6	#5	#4	#3	#2	#1	#0
4549						FHESP		

[Input type] Parameter input
[Data type] Bit spindle

#2 FHESP Spindle control by Servo/Spindle synchronous control (FSSB type) is:
0: Disabled.
1: Enabled.

> **NOTE**
> 1 When using Servo/Spindle synchronous control (FSSB type), set this parameter to 1.
> 2 In the spindle axis, in which this parameter is 1, FSSB high-speed rigid tapping cannot be used. If using with FSSB high-speed rigid tapping, the alarm (SP9068) “ILLEGAL SPINDLE PARAMETER” is issued.

Alarm and message

Number	Message	Description
PW0037	SV/SP COMBINATION ERROR	- The servo axis has dummy axis setting. Please check parameters (No.1023, bit 0 of No.2009, and bit 4 of No.11802). - The setting for the function for spindle axis number to synchronize with servo axis is illegal. Please check parameters (bit 0 of No.3716, No.3717, and No.24204).
SV0417	ILLEGAL SERVO PARAMETER	- The setting for the function, which is not available with Servo/Spindle synchronous control (FSSB type) together, is done. Please turn it off.
SP9068	ILLEGAL SPINDLE PARAMETER	- Servo/Spindle synchronous control (FSSB type) and FSSB high-speed rigid tapping cannot be used together. Please check the parameter (bit 1 and bit2 of No.4549). - In the case of Servo/Spindle synchronous control (FSSB type) using, please turn off FSSB high-speed rigid tapping.

Diagnose

1612	The index number of the spindle axis that synchronizes with each servo axis

[Data Type] Byte axis

The index number of the spindle axis that can use direct communication between a spindle amplifier and a servo amplifier on FSSB connection is displayed.

10.22.3 Phase Synchronization for Servo/Spindle Synchronous Control

Outline

There are three functions on Servo/Spindle synchronous control.

(a) Servo motor spindle control
Can rotate the servo motor at the rotation speed specified with an input signal.

(b) Servo motor spindle synchronization
Can rotate the servo motor in synchronization with the feedback pulses from the spindle sensor.

(c) Differential speed synchronization
  (i) Can superimpose a command from the CNC on the servo motor in servo motor spindle synchronization.
  (ii) Can superimpose the rotation speed specified with an input signal on the servo motor in servo motor spindle synchronization.

By this function, the phase of sub-spindle (the servo motor machine coordinate zero point in the status of reference position establishment) can match the phase of main-spindle (the spindle sensor position determined by the one-rotation signal) on (b) Servo motor spindle synchronization.
Addition, the phase can be shifted by the parameter.

Refer to the manual CONNECTION MANUAL (FUNCTION) (B-64693EN-1) "11.22.1 Servo/Spindle synchronous control" for the basic specification of Servo/Spindle synchronous control.

It is necessary to set Servo/Spindle synchronous control (FSSB type). Refer to the manual CONNECTION MANUAL (FUNCTION) (B-64693EN-1) "11.22.2 Servo/Spindle synchronous control (FSSB type)" for details.

Enable spindle synchronous control (bit 4 (SYC) of parameter No.8133 is 1) for this function.

Explanation

The phase of sub-spindle (the servo motor machine coordinate zero point in the status of reference position establishment) can match the phase of main-spindle (the spindle sensor position determined by the one-rotation signal). Addition, the phase can be shifted by the parameter No.3845.
The method of phase synchronization is shown as follows.

(1) Establish the reference position of sub-spindle (servo motor) by Reference position establishment. (Reference position establishment signals ZRFx <Fn120.x> is "1")
(2) Execute spindle orientation of main-spindle (spindle motor).
(3) When Servo motor spindle synchronization start signal SYSS <Gn061.2> is set "1", it becomes servo motor spindle synchronization mode.
(4) When it becomes servo motor spindle synchronization mode, Servo motor spindle synchronization mode signal SYSSM <Fn090.5> becomes "1".
(5) When it becomes servo motor spindle synchronization mode, sub-spindle accelerates or decelerates to the rotation speed of main-spindle.
(6) When the acceleration/deceleration of sub-spindle is completed, Servo motor spindle synchronization mode acceleration/deceleration completion signal SYAR <Fn090.4> becomes "1".

(7) When Phase synchronization for Servo/Spindle synchronous start signal SYPST <Gn517.7> is set "1" after the acceleration/deceleration of sub-spindle is completed, phase synchronization starts.
(8) The shift value between main-spindle and sub-spindle can be set by parameter No.3845.
(9) It accelerates or decelerates from the current spindle motor speed to the setting speed of parameter No.3846 by Linear acceleration/deceleration with the acceleration rate based on parameter No.3843.
(10) When the synchronization error pulse value become small value more than allowable error pulse value set by parameter No.3847, Phase synchronization for Servo/Spindle synchronous finished signal SYPFN <Fn527.6> becomes "1".
(11) After Phase synchronization for Servo/Spindle synchronous finished signal SYPFN <Fn527.6> becomes "1", set "0" to Phase synchronization for Servo/Spindle synchronous start signal SYPST <Gn517.7>.
(12) When Servo motor spindle synchronization start signal SYSS <Gn061.2> is set "0", servo motor spindle synchronization mode is canceled.
(13) Servo motor spindle synchronization mode acceleration/deceleration completion signal SYAR <Fn090.4> and Phase synchronization for Servo/Spindle synchronous finished signal SYPFN <Fn527.6> become "0".
(14) Sub-spindle decelerates and stops.
(15) When the acceleration/deceleration of sub-spindle is completed, Servo motor spindle synchronization mode signal SYSSM <Fn090.5> becomes "0".

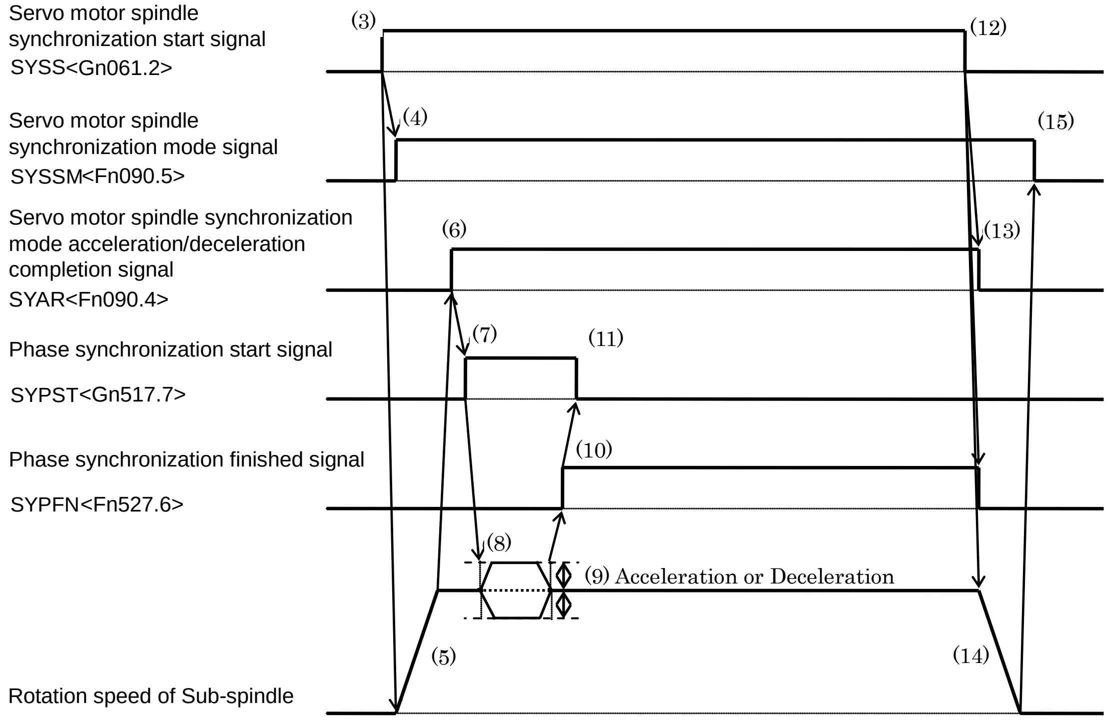


Fig. 10.22.3 Signal sequence of phase synchronization on Servo/Spindle synchronous control

NOTE

NOTE

1 Phase synchronization cannot be executed in superimposition by differential speed synchronization or motion command. Addition, if differential speed synchronization or motion command is specified in phase synchronization, phase synchronization is canceled and differential speed synchronization or motion command is executed. In this case, Phase synchronization for Servo/Spindle synchronous error signal SYPER <Fn527.7> becomes "1".

2 Phase synchronization cannot be executed while canceling Servo/Spindle synchronous control.
In addition, If Servo/Spindle synchronous control is canceled during execution phase synchronization, phase synchronization is canceled.

3 If the servo software or the spindle software doesn’t correspond to this function, Phase synchronization for Servo/Spindle synchronous error signal SYPER <Fn527.7> becomes "1" when Phase synchronization for Servo/Spindle synchronous start signal SYPST <Gn517.7> is set "1".

4 Depending on the mechanical rigidity or the servo response, the synchronization error might occur in high-speed rotation or acceleration/deceleration. Therefore, set big value more than the synchronization error to allowable error pulse value on phase synchronization (parameter No.3847).
Addition, using together with "Soft start/stop function" of spindle motor is effective for a decrease in synchronization error at acceleration/deceleration. Refer to Spindle motor parameter manual (B-65280EN) for details.

5 On phase synchronization, the setting of gear rate is as following.
- Main-spindle (spindle motor)
  The gear ratio between main-spindle and spindle sensor can be used only 1:1.
  On the system that the arbitrary gear ratio between motor sensor and spindle (parameter Nos.4171-4174) or the arbitrary gear ratio between spindle sensor and spindle (parameter Nos.4500-4503) is used, phase synchronization cannot be used.
- Sub-spindle (servo motor)
  The gear rate between sub-spindle and servo motor can be set n:m.

6 Always use Feed-forward function of sub-spindle (servo motor). It is recommended to set 100% to feed-forward coefficient. Refer to "Feed-forward Function" of Servo motor parameter manual (B-65270EN).

7 It is necessary to set Servo/Spindle synchronous control (FSSB type) for this function. Refer to the manual CONNECTION MANUAL (FUNCTION) (B-64693EN-1) "11.22.2 Servo/Spindle synchronous control (FSSB type)" for details.

8 It is necessary to establish the reference position (Reference position establishment signals ZRFx <Fn120.x> is "1") of sub-spindle (servo motor) before phase synchronization start from power supply turning on.
However, if the combination between main-spindle (spindle motor) and sub-spindle (servo motor) is not changed, the reference position establishment since the second times is unnecessary.

9 Set rotary axis A type (bit 0 (ROT) of parameter No.1006 is set 1 and bit 1 (ROS) of parameter No.1006 is set 0) to sub-spindle (servo motor).

NOTE

10 When Servo/Spindle synchronous control is executed, the relationships between coordinates and actual machine position of sub-spindle (servo motor). Thus, the base phase of sub-spindle (servo motor) is the machine coordinate zero point in the status of reference position establishment.

11 Other notes follow notes of Servo/Spindle synchronous control written in the following manual.
- CONNECTION MANUAL (FUNCTION) (B-64693EN-1) "11.22.1 Servo/Spindle synchronous control"

⚠ **CAUTION**

1 Set "0" to Phase synchronization for Servo/Spindle synchronous start signal SYPST <Gn517.7>, after Phase synchronization for Servo/Spindle synchronous finished signal SYPFN <Fn527.6> becomes "1".

2 Phase synchronization cannot execute when Phase synchronization for Servo/Spindle synchronous error signal SYPER <Fn527.7> is "1". Phase synchronization for Servo/Spindle synchronous error signal SYPER <Fn527.7> becomes "0", when Phase synchronization for Servo/Spindle synchronous start signal SYPST <Gn517.7> is set "0".

3 When a motor excitation is cut (Emergency stop, Servo/Spindle alarm is issued, or etc...), set "0" to Phase synchronization for Servo/Spindle synchronous start signal SYPST <Gn517.7>.

4 Set a multiple of 4.096 to the shift amount per one rotation (parameter No.1260) of sub-spindle (servo motor).

5 If the gear ratio for sub-spindle and servo motor is not integer and these connect is semi-closed system, the phase of sub-spindle when the power supply is turned on again will not be guaranteed.

Signal

Phase synchronization for Servo/Spindle synchronous start signal SYPST <Gn517.7>

[Classification] Input signal

[Function] This signal starts phase synchronization for Servo/Spindle synchronous.

[Operation] On Servo/Spindle synchronous, when this signal set “1” from “0”, phase synchronization starts.

Phase synchronization for Servo/Spindle synchronous finished signal SYPFN <Fn527.6>

[Classification] Output signal

[Function] This signal notifies that Phase synchronization for Servo/Spindle synchronous completed.

[Operation] This signal becomes “1” in the following case:
- When the synchronization error pulse value become small value more than allowable error pulse value after phase synchronization was executed.

This signal becomes “0” in the following cases:
- When Servo/Spindle synchronous mode is off.
- When the synchronization error pulse value become big value more than allowable error pulse value in Servo/Spindle synchronous mode.
- When Phase synchronization for Servo/Spindle synchronous error signal SYPER becomes “1”.

Phase synchronization for Servo/Spindle synchronous error signal SYPER <Fn527.7>

[Classification] Output signal

[Function] This signal notifies that Phase synchronization for Servo/Spindle synchronous cannot complete.

[Operation] This signal becomes “1” in the following cases:
- When the setting of phase synchronization for Servo/Spindle synchronous is incorrect.
- When phase synchronization cannot start though Phase synchronization for Servo/Spindle synchronous start signal SYPST is set “1”. (When the feedrate during phase synchronization (parameter No.3846) is 0, when the base position of phase synchronization is not detected, or etc…)
- When phase synchronization is interrupted in execution of phase synchronization. (When differential speed synchronization is specified in execution of phase synchronization, or etc…)
- When Phase synchronization for Servo/Spindle synchronous start signal SYPST is set “1” in execution of differential speed synchronization.
- When the servo software or the spindle software doesn’t correspond to this function.

This signal becomes “0” in the following cases:
- When Servo/Spindle synchronous is mode off.
- When Phase synchronization for Servo/Spindle synchronous start signal SYPST is set “0”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn517	SYPST							

	#7	#6	#5	#4	#3	#2	#1	#0
Fn527	SYPER	SYPFN						

Parameter

3845	Shift value of phase synchronization for Servo/Spindle synchronous control

[Input type] Parameter input
[Data type] Word path
[Unit of data] Detection unit
[Valid data range] 0 to 4095

Set a shift value of phase synchronization for Servo/Spindle synchronous control to this parameter. One rotation of spindle is 4096 pulses.

Example) If the sub-spindle should be at a quarter rotation position from main-spindle, set 1024 to this parameter.

3846	Feedrate during phase synchronization for Servo/Spindle synchronous control

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg/min (machine unit)
[Valid data range] Refer to standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set a feedrate during phase synchronization for Servo/Spindle synchronous control to this parameter.

It accelerates or decelerates from the current spindle motor speed for this parameter by linear acceleration/deceleration with the acceleration rate based on parameter No.3843.

Example) When the increment system IS-B and CMR (parameter No.1820) is 1, the amount of movement will be 4.096[deg] per rotation of the spindle. Therefore, if speed during phase synchronization will be 10[min^{-1}], set 40.96[deg/min] to this parameter.

> **NOTE**
> Set this parameter when Phase synchronization start signal SYPST is "0".
> This parameter setting is reflected when Phase synchronization start signal SYPST become "1" from "0".

3847	Allowable error pulse value on phase synchronization for Servo/Spindle synchronous control

[Input type] Parameter input
[Data type] Word path
[Unit of data] Detection unit
[Valid data range] 0 to 4095

On phase synchronization for Servo/Spindle synchronous control, when the synchronization error pulse value between main-spindle and sub-spindle become small value more than this parameter, Phase synchronization finished signal SYPFN become "1".

> **NOTE**
> Depending on the mechanical rigidity or the servo response, the synchronization error might occur in high-speed rotation or acceleration/deceleration. Therefore, set big value more than the synchronization error to this parameter.

Reference item

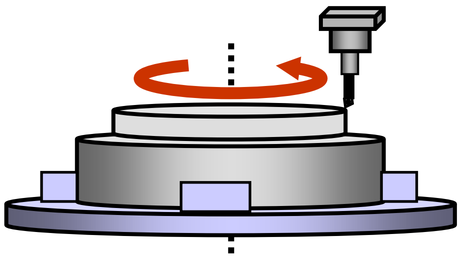
Manual name	Item name
FANUC Series 0*i*-F Plus CONNECTION MANUAL (FUNCTION) (B-64693EN-1)	11.22.1 Servo/Spindle synchronous control
	11.22.2 Servo/Spindle synchronous control (FSSB type)
FANUC AC SPINDLE MOTOR α*i* series / β*i* series FANUC BUILT-IN SPINDLE MOTOR B*i* series PARAMETER MANUAL (B-65280EN)	Soft start/stop function
FANUC AC SERVO MOTOR α*i* series / β*i* series FANUC LINEAR MOTOR L*i*S series FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR D*i*S series PARAMETER MANUAL (B-65270EN)	Feed-forward function

10.23 HIGH-PRECISION SPINDLE SPEED CONTROL

Overview

A large-scale machine cuts a large-diameter workpiece by turning it. Conventional spindle commands (S code commands) cannot specify a surface speed (cutting speed) for large-diameter workpieces in detail because they specify the spindle speed using an integer value. This function uses numbers having decimal points, thus enabling the cutting speed to be specified in detail.

Cutting a large-diameter workpiece by turning it



Explanation

Setting bit 3 (SDP) of parameter No. 3798 to 1 enables high-precision spindle speed control. (Resetting bit 3 (SDP) of parameter No. 3798 to 0 enables conventional spindle commands using integers.) When high-precision spindle control is in use, spindle speed commands (S code commands, a command for maximum spindle speed clamping (G50), and a constant-surface speed control command (G96)) can be issued in units of 0.1 min^{-1} (for constant-surface speed control, 0.1 m/min).

<Command format >
S code command:
M03 S3.5 (Making the spindle rotate at 3.5 [min^{-1}])
Example)
Using high-precision spindle speed control enables the following surface speeds to be selected for a workpiece 4 meters in diameter.

Spindle command	Spindle speed[min^{-1}]	Surface speed [m/min]
S3 M03 ;	3	37.7
S3.2 M03 ;	3.2	40.2
S3.5 M03 ;	3.5	44.0
S3.8 M03 ;	3.8	47.8
S4 M03 ;	4	50.3

Command for maximum spindle speed clamping (G50):
G50 S3.5 (Clamping the spindle speed at 3.5 [min^{-1}])

Constant surface speed control (G96) :
G96 S44.5 (Making the spindle rotate at a surface speed of 44.5 [m/min])
Example)
Using high-precision spindle speed control makes the spindle rotate at the speeds listed below when the specified surface speed is 44.5 [m/min] or 45 [m/min].

Reference-axis position	Surface speed 44.5[m/min]	Surface speed 45[m/min]
	Spindle speed [min^{-1}]	
X=0.1[m];	70.8	71.6
X=0.5[m];	14.2	14.3
X=1.0[m];	7.1	7.2
X=1.5[m];	4.7	4.8
X=2.0[m];	3.5	3.6

The range of speeds specified with axis rotation commands (S code commands) is as follows:

- For serial spindle
  0.0 to 1000.0[min^{-1}]
- For analog spindle
  0.0 to 400.0[min^{-1}]
- For servo motor (using spindle control with servo motor)
  0.0 to 1000.0[min^{-1}]

> **NOTE**
> Using spindle rotation of a speed control type (bit 0 (SPCx) of parameter No. 11000 = 1) requires a servo motor/spindle configuration having a servo motor-to-spindle speed reduction ratio of not higher than 1/40.

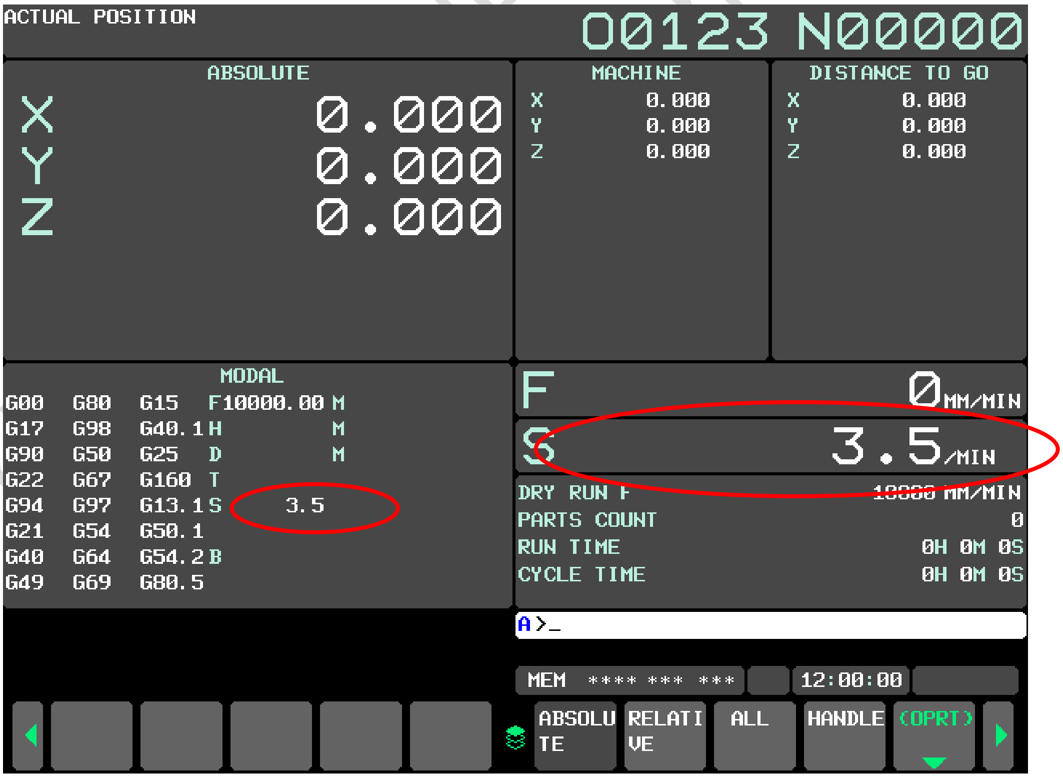
An integer command is assumed if the command specifies a value having no decimal point. For example, if S55 is issued, S55.0 is assumed.

Enabling high-precision spindle speed control makes changes to the following parameter settings and the increment system of data as follows:

- The settings of parameter No. 3771 (minimum spindle speed in the constant-surface speed control mode (G96)) and parameter No. 3772 (upper limit to the rotation speed of individual spindles) are integer data in units of 0.1 min^{-1}.
- The setting of parameter No. 3732 (spindle speed in spindle orientation or spindle motor speed in spindle gear shifting) is integer data in units of 0.1 min^{-1}.
- Data related to spindle speeds in spindle speed signals and values in spindle function code signals are in units of 0.1 min^{-1}.
- Actual spindle speeds read by applications (such as macros, C Language Executor, and FOCAS2) are integer data in units of 0.1 min^{-1}.

Actual spindle speed and S code command displays

If high-precision spindle speed control is enabled, the actual spindle speed and S code commands are displayed using a decimal point.



NOTE

1. When high-precision spindle speed control is enabled, ******.* is displayed if the actual spindle speed or an S command exceeds a maximum allowable display digits.
2. The following screens do not support a display with a decimal point for the actual spindle speed and the S code commands. If high-precision spindle speed control is enabled, the fractional part is rounded off.
   - Operating monitor screen
   - Dynamic graphic (tool path screen)
   - Dynamic graphic (animation drawing screen)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3798				SSI	SDP			

[Input type] Parameter input
[Data type] Bit

#3 SDP High-precision spindle speed control is:
0: Not used.
1: Used.

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#4 SSI The resolution enabled for the spindle speed command is:
0: Maximum spindle speed/4095 [min^{-1}].
1: Maximum spindle speed/16383 [min^{-1}].

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

	#7	#6	#5	#4	#3	#2	#1	#0
4809								NSY

[Input type] Parameter input
[Data type] Bit

T

#0 NSY When the spindle speed changes during spindle synchronization control, resolution improvement is:
0: Disabled. (Speed change in units of about 3.7 [min^{-1}])
1: Enabled. (Speed change in units of about 0.03 [min^{-1}] at minimum but not higher than maximum spindle speed/4095 [min^{-1}])

This parameter is valid when spindle synchronization control or spindle-spindle polygon turning is used.
Using high-precision spindle speed control and spindle synchronization control simultaneously requires setting the parameter to 1.

NOTE
1 When this parameter is set, the power must be turned off before operation is continued.
2 Using this function requires the serial spindle software that supports it.

M

#0 NSY When the spindle speed changes during spindle synchronization control, resolution improvement is:
0: Disabled. (Speed change in units of about 3.7 [min^{-1}])
1: Enabled. (Speed change in units of about 0.03 [min^{-1}] at minimum but not higher than maximum spindle speed/4095 [min^{-1}])

This parameter is valid when spindle synchronization control is used.
Using high-precision spindle speed control and spindle synchronization control simultaneously requires setting the parameter to 1.

NOTE
1 When this parameter is set, the power must be turned off before operation is continued.
2 Using this function requires the serial spindle software that supports it.

	#7	#6	#5	#4	#3	#2	#1	#0
11000								SPCx

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 SPCx SV speed control mode of spindle control with servo motor is:
0: Position control type.
1: Velocity control type.

Notes

- The speed of the feed axis may vary during feed per revolution at a rotation speed of 1 min^{-1} or lower depending on the detection unit used for the feedback data (4096 (pulse/rev)) from the position coder.
- Changing bit 3 (SDP) of parameter No. 3798 enables this function only after the power is turned off and on again.
- If a decimal point is omitted, the desktop calculator decimal point setting is assumed regardless of the setting of bit 0 (DPI) of parameter No. 3401.
  Example: If S123 is issued, S123.0 is assumed.
- If a command with two or more digits after the decimal point specified is issued, the specified value is rounded off to one digit after the decimal point.
  Example: If S123.45 is issued, S123.5 is assumed.
- In S commands that call a subprogram using an S code, the number of digits after the decimal point is not a target with respect to the number of permissible S code digits (parameter No. 3031).

Limitation

- If an analog spindle motor is used, the spindle may fail to rotate at a specified speed depending on the type of the spindle motor and the resolution of the spindle motor drive amplifier with respect to the speed command voltage, and the spindle motor-to-spindle speed reduction ratio.
- High-precision spindle speed control is invalid for rigid tapping S commands. If a command with a decimal point is issued, the command is assumed to specify an integer (with the fraction rounded off).
- The M type gear selection method is unusable.
- The Manual Guide *i* is unusable.

10.24 SIMPLE SPINDLE ELECTRONIC GEAR BOX

Overview

This function executes spindle synchronous control between two serial spindles based on input signals so that one spindle (slave spindle) follows the other spindle (master spindle). It uses a method of referencing directly feedback pulses to enable the slave spindle to follow fluctuations in the master spindle speed with a small error, thereby achieving high-precision spindle synchronous control.

An example of using the function might be rotary guide bush control where two spindles are used.

An electronic gear box (hereafter called EGB) for the spindle is used to make the two spindles synchronize with each other.

Using the function requires enabling the spindle serial output (bit 5 (SSN) of parameter No.8133 is 0) and the Cs contour control (bit 2 (SCS) of parameter No.8133 is 1).

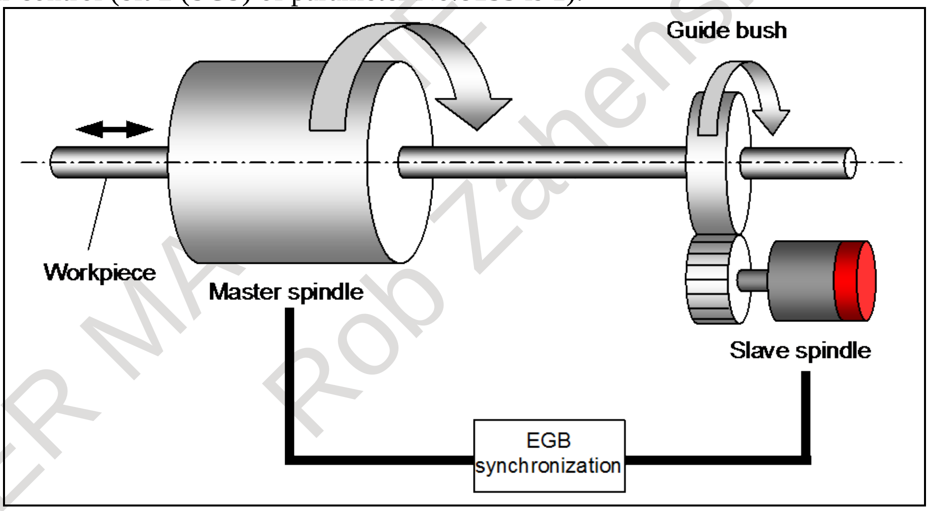


Fig. 10.24 (a) Application example

Specification

With the spindle EGB, the synchronous pulse is produced from the feedback pulse output by the position detector attached to the master axis, and the slave axis rotates according to the synchronous pulse. The feedback pulse sent from master spindle to the slave spindle is forwarded by communication between spindle amplifiers.

This function uses the PMC input signal to turn the simple spindle EGB mode ON/OFF.

Turning on the simple spindle EGB mode ON does not impose any limitation to the spindle control mode for the master spindle. However, the slave spindle must be in the Cs contour control mode.

During the simple spindle EGB mode, the coordinates of the slave spindle are not updated even if the slave spindle moves in synchronization with the master spindle.

The simple spindle EGB needs the same spindle system configuration (motor, amplifier, and sensor) as the spindle EGB. For details, see the Item, “Spindle EGB (spindle electronic gear box)” in the FANUC

AC SPINDLE MOTOR α*i* series, FANUC AC SPINDLE MOTOR β*i* series, FANUC BUILT-IN SPINDLE MOTOR bi series PARAMETER MANUAL (B-65280EN).

Differences between the simple spindle EGB and the spindle EGB

The following table lists the differences between the simple spindle EGB and the spindle EGB.

Table 10.24 Differences between the simple spindle EGB and the spindle EGB

	Simple spindle electronic gear box	Spindle electronic gear box
Feature	- PMC input signal is used to turn synchronization ON/OFF. - No positional control is performed during the synchronization mode.	- G code command is used to turn synchronization ON/OFF. - Positional control is performed during the synchronization mode. - Gear processing functions such as helical gear compensation and retract are usable.
Major use	For rotary guide bush, which requires only simple spindle synchronization rather than positional control	For gear processing, which requires positional control as well as retract operations

Signal

Simple spindle EGB signals

SSEGB1<Gn351.0>: First spindle
SSEGB2<Gn351.1>: Second spindle
SSEGB3<Gn351.2>: Third spindle
SSEGB4<Gn351.3>: Fourth spindle

[Classification] Input signal

[Function] Select the simple spindle EGB.

[Operation] When each signal becomes "1", the simple spindle EGB mode is turned ON with the respective serial spindles selected as a slave.
When each signal becomes "0", the simple spindle EGB mode is turned OFF for the respective serial spindles being used as a slave.

> **NOTE**
> 1 These signals are valid when the bit 0 (SEGs) of parameter No. 7705 is 1.
> 2 If the simple spindle EGB mode is turned ON or OFF when both the master and slave spindles are not at halt, the slave spindle accelerates or decelerates abruptly. When turning the simple spindle EGB mode ON or OFF, be sure to previously put both the master and the slave spindles at halt.
> 3 During the simple spindle EGB mode, keep the slave spindle in the Cs contour control mode.

Simple spindle EGB mode signals

SSEGBM1<Fn351.0>: First spindle
SSEGBM2<Fn351.1>: Second spindle
SSEGBM3<Fn351.2>: Third spindle
SSEGBM4<Fn351.3>: Fourth spindle

[Classification] Output signal

[Function] These signals indicate whether the simple spindle EGB mode is ON for the respective spindles selected as a slave spindle.

[Output cond.] Each signal becomes "1" when:

- The simple spindle EGB mode is ON for the respective spindles selected as a slave spindle.

Each signal becomes "0" when:

- The simple spindle EGB mode is OFF for the respective spindles selected as a slave spindle.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn351					SSEGB4	SSEGB3	SSEGB2	SSEGB1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn351					SSEGBM4	SSEGBM3	SSEGBM2	SSEGBM1

Sequence

- Simple spindle EGB mode ON

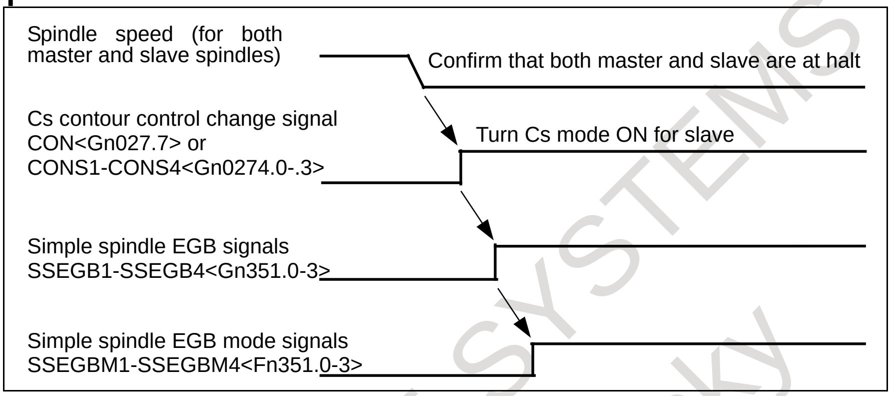


Fig. 10.24 (b) Simple spindle EGB mode ON

- Simple spindle EGB mode OFF

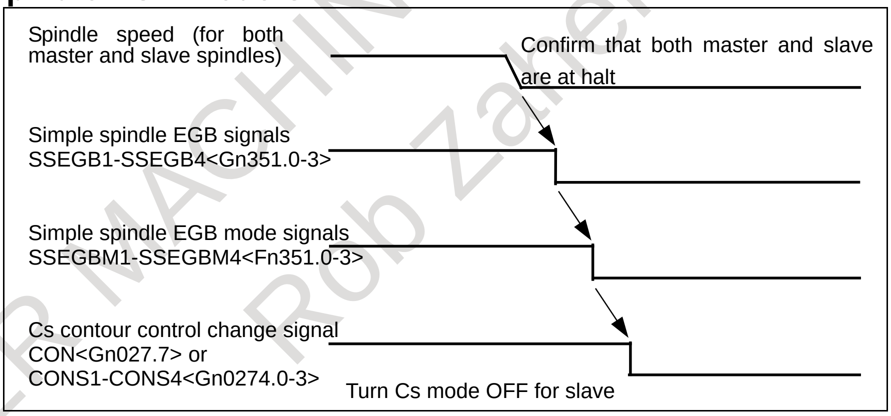


Fig. 10.24 (c) Simple spindle EGB mode OFF

NOTE

1. When turning the simple spindle EGB mode ON or OFF, keep both the master and slave spindles at halt.
2. During the simple spindle EGB mode, the positional deviation of the Cs axis for the slave spindle from the master spindle is returned to the slave spindle. For the slave spindle in the simple spindle EGB mode, checks are constantly made for any excess error value during movement.

NOTE

3 During the simple spindle EGB mode, the coordinates of the Cs axis for the slave spindle are not updated even if the slave spindle is synchronized with the master spindle.
4 During the simple spindle EGB mode, it is impossible to make a reference point return for the Cs axis for the slave spindle.
5 During the simple spindle EGB mode, it is impossible to perform composite control for the Cs axis for the slave spindle.
6 The simple spindle EGB needs the same spindle system configuration (motor, amplifier, and sensor) as the spindle EGB.
7 The synchronization mode is canceled if a servo alarm or spindle alarm condition is detected.
8 Making a request for an emergency stop during synchronization causes both the master and slave spindles decelerate to stop while they are kept in synchronization. However, they may not stay in synchronization once they stop. Release the slave spindle from the simple EGB mode and Cs mode after an emergency stop.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7705								SEGs

[Input type] Parameter input
[Data type] Bit spindle

#0 SEGs Simple spindle EGB function is:
0: Not used.
1: Used.
Set 1 for a serial spindle used as the slave axis for the simple spindle EGB function,.

	#7	#6	#5	#4	#3	#2	#1	#0
4395						SSE		

[Input type] Parameter input
[Data type] Bit spindle

#2 SSE The simple spindle EGB function is:
0: Disabled.
1: Enabled.
Set this parameter to 1 for the master and slave spindles together.

	#7	#6	#5	#4	#3	#2	#1	#0
4352	SEM	SES		FFALWS				

[Input type] Parameter input
[Data type] Bit spindle

#4 FFALWS Feed-forward setting is enabled:
0: Only during cutting feed.
1: Always.
Set this parameter 1 for the slave spindle.

#6 SES The spindle EGB function for the slave spindle is:
0: Disabled.
1: Enabled.
Set this parameter 1 for the slave spindle.

#6 SEM The spindle EGB function for the master spindle is:
0: Disabled.
1: Enabled.
Set this parameter 1 for the master spindle.

4036	Feed forward coefficient

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%
[Valid data range] 0 to 10000
The feed forward coefficient for Cs axis contour control is set as follows:
Setting ≤ 100: In units of 1%
Setting > 100: In units of 0.01%
Set this parameter to 100 for the slave spindle.

4386	Number of sinusoidal waves from the master spindle position detector

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1λ/rev
[Valid data range] 0, 64 to 4096
Specify the number of sinusoidal waves per spindle revolution received from the master spindle position detector.
Use this parameter for the slave spindle amplifier.
If the parameter is 0, the synchronous ratio assumed to be 0.

4387	Numerator of synchronization coefficient

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] -32767 to 32767
Set a synchronous coefficient numerator for the slave spindle.

4388	Denominator of synchronization coefficient

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 1 to 32767
Set a synchronous coefficient denominator for the slave spindle.

Reference item

Manual name	Item name
FANUC AC SPINDLE MOTOR α*i* series FANUC AC SPINDLE MOTOR β*i* series FANUC BUILT-IN SPINDLE MOTOR β*i* series PARAMETER MANUAL (B-65280EN)	Spindle EGB (Spindle electronic gear box)

10.25 SPINDLE SPEED COMMAND CLAMP

Overview

By using this function, spindle speed can be clamped to the maximum speed by setting on internal relay (R signal) data.

Explanation

The internal relay addresses (R signal) is specified with parameter No.3773 for each spindle. Four bytes starting at the setting are used for each spindle. Spindle speed is clamped to the maximum speed by setting on internal relay (R signal) data.

For example, this function can be used to change maximum speed depending on selected tool as following procedure.

1. On the PMC side, the maximum speed depending on the selected tool is set on the internal relay addresses (R signal) specified with the parameter. (The PMC program is created by machine tool builders)
2. The CNC reads maximum speed from PMC internal relay (R signal). The CNC clamp spindle speed to maximum speed immediately if the spindle speed exceeds the maximum speed.

Figure 11.26(a) show the processing of the spindle speed clamp when a command is specified after the tool change.

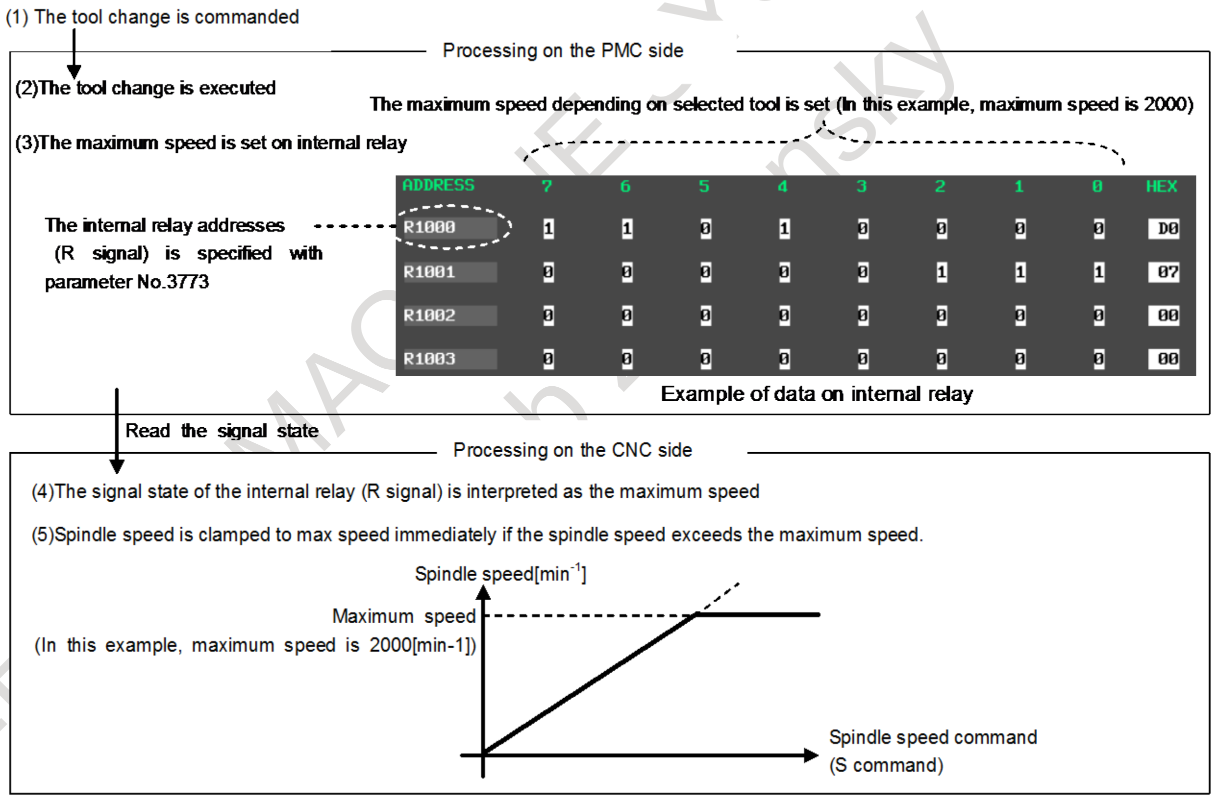


Fig.10.25 (a) processing of this function

> **NOTE**
> 1 This function only effects for a spindle speed command (including spindle override, constant surface speed control), it doesn't effect except spindle speed command (spindle orientation, rigid tapping and so on).
> 2 When spindle speed command control is applied using the PMC, this function has no effect, and the spindle speed is not clamped.

NOTE

3 When the spindle synchronous control is executed, spindle speed of the master and the slave are clamped to the lower value of master's maximum speed and slave's maximum speed.
4 When the M type gear selection method is used, this function has effect even if minimum clamp speed of the spindle motor (parameter No.3735) is larger than the max speed specified on PMC internal relay (R signal).
5 The maximum speed can be set within the range of 0 to 99999999 [min^{-1}]
6 This function has effect for unselected spindle by address P or spindle selection signals SWS1 to SWS4 <Gn027.0 to Gn027.2 and Gn026.3>
7 Depending on bit 3 (SDP) of parameter No. 3798, which is related to High-precision spindle speed control, the maximum speed is changed as following table.

SDP = 0	The signal state of the internal relay (R signal) is interpreted as the maximum speed. The maximum speed can be set within the range of 0 to 99999999 [min^{-1}]
SDP = 1	1/10 of the signal state of the internal relay (R signal) is interpreted as the maximum speed. The maximum speed can be set within the range of 0 to 9999999.9 [min^{-1}]

Parameter

3773	Start address of the R signal specifying maximum speed

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 0 to maximum address (multiple of 4. 0, 4, 8, ...)

This parameter sets the start address of the R signal specifying maximum speed.
Four bytes starting at the setting are used for each spindle.

NOTE

1 As for the setting of parameter
<1> Set a value that is a multiple of 4 (0, 4, 8, etc.).
<2> The range of the R address differs depending on the PMC kind and the memory size. Check the specifications of the PMC, and set a value within the valid range. (Example: R addresses in the range from R0 to R7999 if memory B of the first PMC is used. Thus, values which can be set are 4, 8, 12, 16, …7992, 7996 in this case)
If any setting other than the above items (<1>, <2>) is made, alarm PW5390, "R-ADDRESS SETTING IS ILLEGAL" is issued.
2 When value of parameter No. 3773 is zero, this function has no effect.

> ⚠ **WARNING**
> If an internal relay of the set address in this parameter is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used internal relay.

Alarm and message

Number	Massage	Description
PW5390	R-ADDRESS SETTING IS ILLEGAL	One of the R address range of the PMC set by parameters No. 3773, No. 13541 and No. 13542 or the first address of the range is invalid.

10.26 IMPROVEMENT OF SPINDLE FEEDBACK PULSES

Overview

Feedback pulses of the spindle which is used at the feed per revolution, threading and spindle speed display becomes able to be changed by setting of parameter.
By this function, the fluctuation of axis speed at the feed per revolution and threading is reduced, and a smoother axis moving becomes possible.

Specification

It is usually, feedback pulses of the spindle is 4096 pulses per one rotation of the spindle. However, the setting of feedback pulses of the spindle can be changed by the bit 7 (FBP) of parameter No.3716 and No.3720.

- Feedback pulses per one rotation of the spindle can be chosen from 4096 pulses or $4096 \times 2^{\wedge 14}$ pulses in serial spindle.
- Feedback pulses per one rotation of the spindle can be chosen from 4096 pulses or $4096 \times 2^{\wedge 14}$ pulses in spindle control with servo motor.
- Feedback pulses per one rotation of the spindle can be chosen from 4096 to 400000 pulses in analog spindle.

The changing of feedback pulses of the spindle becomes effective by the setting of bit 7 (FBP) of parameter No.3716. And number of feedback pulses is set by parameter No.3720. However, when bit 7 (FBP) of parameter No.3716 is set to 1, it is not necessary to set parameter No.3720 because parameter No.3720 is set automatically in serial spindle and spindle control with servo motor.

> **NOTE**
> 1 When feedback pulses in analog spindle are not 4096pulse/rev, alarm SP1255, "CAN NOT CHANGE SPINDLE MODE" occurs if rigid tapping or spindle positioning is specified. (This alarm not occurs when serial spindle.)
> 2 The effect of this function depends on the resolution of detector.
> (In serial spindle, the resolution of each detector is as follows. This function is not effective in αi position coder.
>
> | | |
> |---|---|
> | αiBZ sensor | 360000pulse/rev |
> | αiCZ sensor | 3600000pulse/rev |
> | αi position coder | 4096pulse/rev |
>
> In spindle control with servo motor, this function is not effective as long as the resolution of detector is not small larger than 4096 pulses per one rotation.)
> Moreover, the accuracy is not assured by the above values because they are indicated as the resolution of detector. Select the detector with enough accuracy for machining accuracy.

> **NOTE**
> 3 Maximum spindle speed becomes as follows when this function is effective.
> In serial spindle or spindle control with servo motor, maximum spindle speed is 99999 min^{-1}.
> However, maximum spindle speed depends on the specification of spindle motor, spindle amplifier and detector too.
> In analog spindle, maximum spindle speed depends on the time rules of interface of position coder.
> Refer “Analog spindle connection material” (A-65618EN) about the time rules of interface of position coder.
> About the applicable pulse number of position coder and upper limit of spindle speed, the influence such as the machine vibrations is considered, calculate the number in which the margin is given for "time rules" satisfy without fail in any case.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3716	FBP							

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit spindle

#7 FBP Changing the number of spindle position coder pulses is
0: Not executed.
1: Executed.

3720	Number of position coder pulses

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Number of pulses
[Valid data range] Serial spindle
: 0, 4096 When bit 7 (FBP) of parameter No.3716 is set to 1: 4096×2^{14}
Spindle control with servo motor
: 0, 4096 When bit 7 (FBP) of parameter No.3716 is set to 1: 4096×2^{14}
Analog spindle
: 0, 4096 When bit 7 (FBP) of parameter No.3716 is set to 1: from 0 to 400000

Set the number of position coder pulses.
In analog spindle, 4096 is set to parameter automatically if parameter No.3720 is set to 0 or less.
In serial spindle or spindle control with servo motor, the setting method is changed by setting of the bit 7 (FBP) of parameter No.3716.

- When the bit 7 (FBP) of parameter No.3716 is set to 0:
  4096 is set to parameter automatically if parameter No.3720 is set to 0 or less.
- When the bit 7 (FBP) of parameter No.3716 is set to 1:
  $4096 \times 2^{\wedge 14}$ is set to parameter automatically.

Alarm and message

Number	Message	Description
SP1255	CAN NOT CHANGE SPINDLE MODE	In the analog spindle control, rigid tapping and spindle positioning can not be used except when the number of pulses output from the position coder of the spindle is 4096pulse/rev.

10.27 RESOLUTION OF SPINDLE SPEED COMMAND

Overview

By this function, the resolution of spindle speed commands and related signals is improved, and high accuracy spindle speed command is achieved.
This function is effective on serial spindle.

Explanation

When bit 0(SSE) of parameter No.3791 is set to 1, spindle speed command resolution of serial spindle is improved.
The resolution is decided on parameters of Table 10.27 (a).

Table 10.27 (a)

Resolution	Bit 0(SSE) of parameter No.3791	Bit 4(SSI) of parameter No.3798
4096	0	0
16383	0	1
1048575	1	0
1048575	1	1

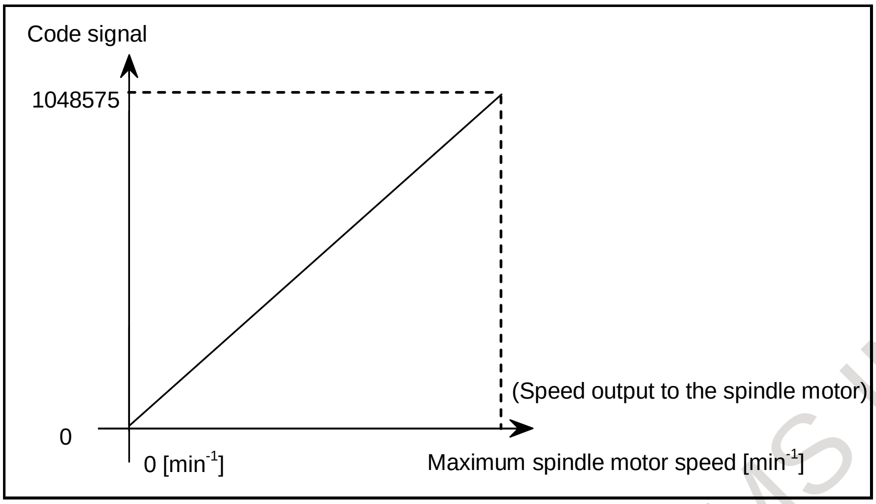
And then, the following signals become high resolution.

- Spindle speed command output

The commanded spindled speed by machining program is calculated through the gear change processing. Refer to chapter of “SPINDLE SPEED CONTROL” in Connection manual (function) (B-64693EN-1) for detail.

When bit 0(SSE) of parameter No.3791 is set to 1, the calculated speed command is outputted by the S32-bit code signals RE01O to RE32O<Fn708 to Fn711> from 0 to 1048575. Then, S12-bit code signals R01O to R12O <Fn036.0 to Fn037.3> are not effective.

$$\text{32-bit code signals output value} = \frac{\text{Spindle motor speed}[\text{min}^{-1}]}{\begin{array}{c}\text{Maximum spindle motor speed}[\text{min}^{-1}]\\ \text{(parameter No.4020)}\end{array}} \times 1048575$$



- Multi spindle control

When Multi spindle control is effective, the speed commands output signals of each spindle are depended the parameter setting in Table 10.27 (b).

Table 10.27 (b)

		Bit 0(SSE) of parameter No.3791 is set to 1
Effective output signals	1st spindle	RE01O to RE32O<Fn708 to Fn711>
	2nd spindle	RE01O2 to RE32O2<Fn712 to Fn715>
	3rd spindle	RE01O3 to RE32O3<Fn716 to Fn719>
	4th spindle	RE01O4 to RE32O4<Fn720 to Fn723>

		Bit 0(SSE) of parameter No.3791 is set to 0	
		BIT 3(MRS) of parameter No.3709 is set to 0	BIT 3(MRS) of parameter No.3709 is set to 1
Effective output signals	1st spindle	R01O to R12O <Fn036.0 to Fn037.3>	R01O to R12O<Fn036.0 to Fn037.3>
	2nd spindle		R01O2 to R12O2<Fn200.0 to Fn201.3>
	3rd spindle		R01O3 to R12O3<Fn204.0 to Fn205.3>
	4th spindle		R01O4 to R12O4<Fn270.0 to Fn271.3>

Refer to Chapter of “Multi spindle control” in “Connection manual (function)” (B-64693EN-1) for detail of spindle select.

NOTE
1. RE01Os to RE32Os are 32-bit, but effective signals are 20-bit.
2. When SSE is set to 1 for a spindle in system, even if common signals setting are set (SSE is 0 and MRS is 0), each spindle signals (R01Os to R12Os) are effective.

- SPINDLE OUTPUT CONTROL BY THE PMC

When bit 0 (SSE) of parameter No.3791 is set to 1, if spindle motor speed is specified from PMC, set signals as follow.

- Switching the controller from the CNC to the PMC, by issuing a signal used to select the spindle motor speed command SINDs
- Setting the spindle motor speed data, calculated by the PMC, in spindle control signals RE01Is toRE12Is. Then, the spindle motor speed command signals R01Is to R12Is is not effective.

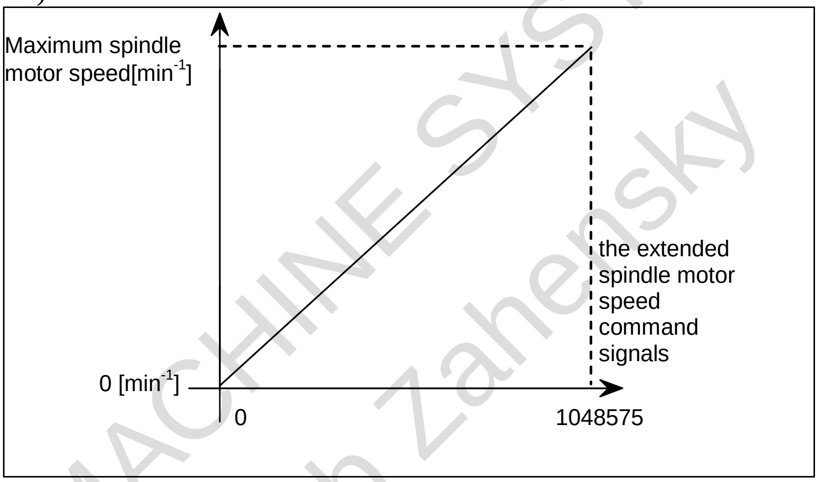
Table 10.27 (c)

		Bit 0(SSE) of parameter No.3791 is set to 0	Bit 0(SSE) of parameter No.3791 is set to 1
Effective spindle motor Speed command signals	**1st spindle**	R01I to R12I<Gn032.0 to Gn033.3>	RE01I to RE32I<Gn708 to Gn711>
	2nd spindle	R01I2 to R12I2<Gn034.0 to Gn035.3>	RE01I2 to RE32I2<Gn712 to Gn715>
	3rd spindle	R01I3 to R12I3<Gn036.0 to Gn037.3>	RE01I3 to RE32I3<Gn716 to Gn719>
	4th spindle	R01I4 to R12I4<Gn272.0 to Gn273.3>	RE01I4 to RE32I4<Gn720 to Gn723>

When bit 0(SSE) of parameter No.3791 is set to 1, a spindle motor speed is set to the data by a following calculating method.

$$\text{Setting value of the extended spindle motor speed command signals} = \frac{\text{Spindle motor speed}[\mathrm{min}^{-1}]}{\begin{array}{c}\text{Maximum spindle motor speed}[\mathrm{min}^{-1}] \\ \text{(parameter No.4020)}\end{array}} \times 1048575$$

(Spindle motor speed)



Following signal that doesn't depend on bit 0(SSE) of parameter No.3791 setting is used on same signals.

Table 10.27 (d)

1st spindle	SIND,SSIN,SGN,<Gn033.7,.6,.5>
2nd spindle	SIND2,SSIN2,SGN2,<Gn035.7,.6,.5>
3rd spindle	SIND3,SSIN3,SGN3,<Gn037.7,.6,.5>
4th spindle	SIND4,SSIN4,SGN4,<Gn273.7,.6,.5>

Refer to Chapter of "Spindle output control by the PMC" in "Connection manual (function)" (B-64693EN-1) for detail of the SINDs, SSINs and SGNs.

- For multi spindle control

When multi spindle control is effective and TYPE-A is selected (bit 2(MSI) of parameter No.3709 is set to 0), the signals for the second to forth spindles RE01I2,3,4 to RE32I2,3,4 are not effective.
Even when multi-spindle control is not effective, the signals for the second to forth spindles RE01I2,3,4 to RE32I2,3,4 can used.

NOTE

Extended spindle motor speed command signals RE01Is to RE32Is are 32-bit, but effective signals are 20-bit.

- Command output to the spindle control unit

The speed command output decided into CNC is commanded to the spindle control unit of serial spindle at digital data following the resolution of Table 10.27 (a).
When bit 0 (SSE) of No.3791 is set to 1, the resolution of digital data is 0 to ±1048575

T

- Actual spindle speed output

When bit 0 (SSE) of No.3791 is set to 1, extended actual spindle speed signals ARE00 to ARE15 <Fn580 to Fn583> can read actual spindle speed.
However, then signals AR00 to AR15<Fn040, Fn041>,AR002 to AR152< Fn202, Fn203>,AR003 to AR153< Fn206, Fn207>,AR004 to AR154< Fn272, Fn273> are not effective.

- For multi spindle control

When the multi spindle control is effective, a actual speed of each spindle is depended on parameter of Table 10.27 (e).

Table 10.27 (e)

		Bit 0(SSE) of parameter No.3791 is set to 0	Bit 0(SSE) of parameter No.3791 is set to 1
Effective Actual spindle speed Output signals	1st spindle	AR00 to AR15<Fn040, Fn041>	ARE00 to ARE31<Fn580 to Fn583>
	2nd spindle	AR002 to AR152< Fn202, Fn203>	ARE002 to ARE312<Fn584 to Fn587>
	3rd spindle	AR003 to AR153< Fn206, Fn207>	ARE003 to ARE313<Fn588 to Fn591>
	4th spindle	AR004 to AR154< Fn272, Fn273>	ARE004 to ARE314<Fn592 to Fn595>

Refer to Chapter of "Multi spindle control" in "Connection manual (function)" (B-64693EN-1) for detail of spindle select.

High-precision spindle speed control

If high-precision spindle speed control is enable (bit 3 (SDP) of parameter No.3798 is set to 1), spindle speed commands (S code commands, a command for maximum spindle speed clamping (G50), and a constant-surface speed control command (G96)) can be issued in units of 0.1 min^{-1} (for constant-surface speed control, 0.1 m/min).Actual spindle speed signals are in units of 0.1 min^{-1}.
Refer to Chapter of "High-precision spindle speed control" in "Connection manual (function)" (B-64693EN-1).

Notes

- The supported serial spindle software is required to this function.
- When signals of multi spindle relation are used, enable multi spindle control function(bit 3 (MSP) of parameter No.8133 is 1) separately.

Limitations

- Analog spindle control

This function is not effective in "Analog spindle control".

- Spindle control with servo motor

This function is not effective in "Spindle control with servo motor".

- Spindle synchronous control / Spindle command synchronous control / Polygon turning with two spindles

Set the same setting at the bit 0 (SSE) of parameter No.3791 when "Spindle synchronous control", "Spindle command synchronous control" or "Polygon turning with two spindles" is used on each spindle.

If the setting is different, the alarm SP1257,” ILLEGAL PARAMETER (No.3791#0)” occurs.

Signal

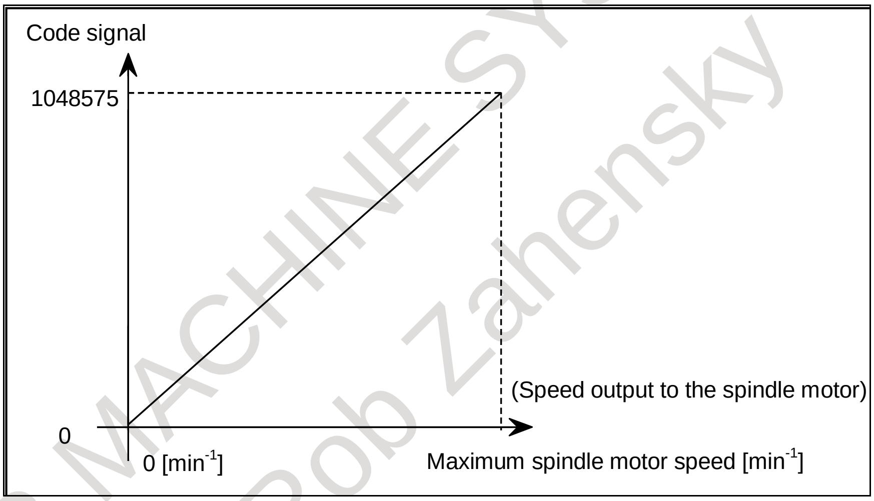
S 32-bit code signals

RE01O to RE32O<Fn708 to Fn711>: for 1st spindle
RE01O2 to RE32O2<Fn712 to Fn715>: for 2nd spindle
RE01O3 to RE32O3<Fn716 to Fn719>: for 3rd spindle
RE01O4 to RE32O4<Fn720 to Fn723>: for 4th spindle

[Classification] Output signal

[Function] When bit 0(SSE) of parameter No.3791 is set to 1, CNC is outputted the speed command output of calculated spindle motor to PMC as the S32-bit code signals RE01Os to RE32Os. Then, S12-bit code signals R01O to R12O<Fn036.0 to Fn037.3> are not effective.
When Multi spindle control is effective and bit 3 (MRS) of parameter No.3709 is set to 1, spindle motor of each spindle speed command are outputted to S32-bit code signals RE01Os to RE32Os. Then, R01Os to R12Os <Fn200.0 to Fn201.3>, <Fn204.0 to Fn205.3>, <Fn270.0 to Fn271.3> are not effective.

[Output cond.] The relationship between the spindle speed command value (calculated by the CNC) and the value output by this signal is as shown below.



> **NOTE**
> S32-bit code signals RE01Os to RE32Os are 32-bit, but effective signals are 20-bit.

Extended spindle motor speed command signals

RE01I to RE32I<Gn708 to Gn711>: for 1st spindle
RE01I2 to RE32I2<Gn712 to Gn715>: for 2nd spindle
RE01I3 to RE32I3<Gn716 to Gn719>: for 3rd spindle
RE01I4 to RE32I4<Gn720 to Gn723>: for 4th spindle

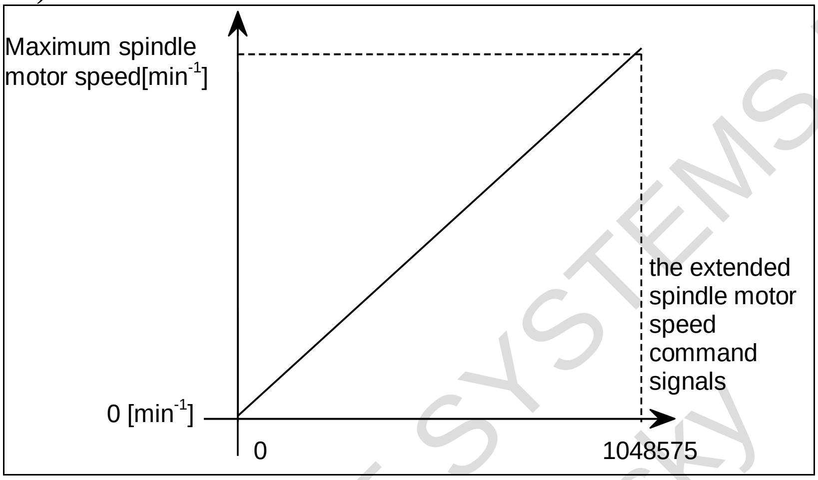
[Classification] Input signal

[Function] When bit 0(SSE) of parameter No.3791 is set to 1, the speed command is possible to the each spindle motor by RE01I to RE32Ix. Then, R01Ix to R12Ix<Gn032.0 to Gn033.3>,<Gn034.0 to Gn035.3>,<Gn036.0 to Gn037.3> , <Gn272.0 to Gn273.3> is not effective.
However, when multi spindle control is effective and TYPE-A is selected (bit 2(MSI) of parameter No.3709 is set to 0), the signals for the second to forth spindles RE01I2,3,4 to RE32I2,3,4 are not effective.

[Operation] If the PMC is being used to control the spindle motor speed command, specify, in binary format, the value obtained using the following expression.

$$\text{Value to be specified by extended spindle motor speed command signals} = \frac{\text{Spindle motor speed}[\min^{-1}]}{\text{Maximum spindle motor speed}[\min^{-1}] \text{ (parameter No.4020)}} \times 1048575$$

(Spindle motor speed)



> **NOTE**
> Extended spindle motor speed command signals RE01Is to RE32Is are 32-bit, but effective signals are 20-bit.

T

Extended actual spindle speed signals

ARE00 to ARE31<Fn580 to Fn583>: for 1st spindle
ARE002 to ARE312<Fn584 to Fn587>: for 2nd spindle
ARE003 to ARE313<Fn588 to Fn591>: for 3rd spindle
ARE004 to ARE314<Fn592 to Fn595>: for 4th spindle

[Classification] Output signal

[Function] When bit 0(SSE) of parameter No.3791 is set to 1, ARE00x to ARE31x can read the actual spindle speed obtained by feedback pulses from the position coder mounted on the spindle. Then, AR00Ox to AR15Ox <Fn202,Fn203>,<Fn206,Fn272>, <Fn202,Fn273> is not effective.
When bit 3 (SDP) of parameter No.3798 is set to 1, Extended actual spindle speed signals are in units of 0.1 $\min^{-1}$. (When bit 3 (SDP) of parameter No.3798 is 0, there are in units of 1 $\min^{-1}$)

[Output cond.] When bit 0(SSE) of parameter No.3791 is set to 1, these 32-bit binary code signals are always output. The actual spindle speed and the signals have the following relationship:

$$Spindle\quad speed = \sum_{i=0}^{31} \left\{ 2^i \times V_i \right\} \times D \min^{-1}$$

Where Vi=0 when AREi is 0
and Vi=1 when AREi is 1
and D=1 when bit 3 (SDP) of parameter No.3798 is 0
and D=0.1 when bit 3 (SDP) of parameter No.3798 is 1

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn708	RE08I	RE07I	RE06I	RE05I	RE04I	RE03I	RE02I	RE01I
Gn709	RE16I	RE15I	RE14I	RE13I	RE12I	RE11I	RE10I	RE09I
Gn710	RE24I	RE23I	RE22I	RE21I	RE20I	RE19I	RE18I	RE17I
Gn711	RE32I	RE31I	RE30I	RE29I	RE28I	RE27I	RE26I	RE25I
Gn712	RE08I2	RE07I2	RE06I2	RE05I2	RE04I2	RE03I2	RE02I2	RE01I2
Gn713	RE16I2	RE15I2	RE14I2	RE13I2	RE12I2	RE11I2	RE10I2	RE09I2
Gn714	RE24I2	RE23I2	RE22I2	RE21I2	RE20I2	RE19I2	RE18I2	RE17I2
Gn715	RE32I2	RE31I2	RE30I2	RE29I2	RE28I2	RE27I2	RE26I2	RE25I2
Gn716	RE08I3	RE07I3	RE06I3	RE05I3	RE04I3	RE03I3	RE02I3	RE01I3
Gn717	RE16I3	RE15I3	RE14I3	RE13I3	RE12I3	RE11I3	RE10I3	RE09I3
Gn718	RE24I3	RE23I3	RE22I3	RE21I3	RE20I3	RE19I3	RE18I3	RE17I3
Gn719	RE32I3	RE31I3	RE30I3	RE29I3	RE28I3	RE27I3	RE26I3	RE25I3
Gn720	RE08I4	RE07I4	RE06I4	RE05I4	RE04I4	RE03I4	RE02I4	RE01I4
Gn721	RE16I4	RE15I4	RE14I4	RE13I4	RE12I4	RE11I4	RE10I4	RE09I4
Gn722	RE24I4	RE23I4	RE22I4	RE21I4	RE20I4	RE19I4	RE18I4	RE17I4
Gn723	RE32I4	RE31I4	RE30I4	RE29I4	RE28I4	RE27I4	RE26I4	RE25I4

	#7	#6	#5	#4	#3	#2	#1	#0
Fn580	ARE07	ARE06	ARE05	ARE04	ARE03	ARE02	ARE01	ARE00
Fn581	ARE15	ARE14	ARE13	ARE12	ARE11	ARE10	ARE09	ARE08
Fn582	ARE23	ARE22	ARE21	ARE32	ARE19	ARE18	ARE17	ARE16
Fn583	ARE31	ARE30	ARE29	ARE28	ARE27	ARE26	ARE25	ARE24
Fn584	ARE072	ARE062	ARE052	ARE042	ARE032	ARE022	ARE012	ARE002
Fn585	ARE152	ARE142	ARE132	ARE122	ARE112	ARE102	ARE092	ARE082
Fn586	ARE232	ARE222	ARE212	ARE322	ARE192	ARE182	ARE172	ARE162
Fn587	ARE312	ARE302	ARE292	ARE282	ARE272	ARE262	ARE252	ARE242
Fn588	ARE073	ARE063	ARE053	ARE043	ARE033	ARE023	ARE013	ARE003
Fn589	ARE153	ARE143	ARE133	ARE123	ARE113	ARE103	ARE093	ARE083
Fn590	ARE233	ARE223	ARE213	ARE323	ARE193	ARE183	ARE173	ARE163
Fn591	ARE313	ARE303	ARE293	ARE283	ARE273	ARE263	ARE253	ARE243
Fn592	ARE074	ARE064	ARE054	ARE044	ARE034	ARE024	ARE014	ARE004
Fn593	ARE154	ARE144	ARE134	ARE124	ARE114	ARE104	ARE094	ARE084
Fn594	ARE234	ARE224	ARE214	ARE324	ARE194	ARE184	ARE174	ARE164

Fn595	ARE314	ARE304	ARE294	ARE284	ARE274	ARE264	ARE254	ARE244

Fn708	RE08O	RE07O	RE06O	RE05O	RE04O	RE03O	RE02O	RE01O
Fn709	RE16O	RE15O	RE14O	RE13O	RE12O	RE11O	RE10O	RE09O
Fn710	RE24O	RE23O	RE22O	RE21O	RE20O	RE19O	RE18O	RE17O
Fn711	RE32O	RE31O	RE30O	RE29O	RE28O	RE27O	RE26O	RE25O
Fn712	RE08O2	RE07O2	RE06O2	RE05O2	RE04O2	RE03O2	RE02O2	RE01O2
Fn713	RE16O2	RE15O2	RE14O2	RE13O2	RE12O2	RE11O2	RE10O2	RE09O2
Fn714	RE24O2	RE23O2	RE22O2	RE21O2	RE20O2	RE19O2	RE18O2	RE17O2
Fn715	RE32O2	RE31O2	RE30O2	RE29O2	RE28O2	RE27O2	RE26O2	RE25O2
Fn716	RE08O3	RE07O3	RE06O3	RE05O3	RE04O3	RE03O3	RE02O3	RE01O3
Fn717	RE16O3	RE15O3	RE14O3	RE13O3	RE12O3	RE11O3	RE10O3	RE09O3
Fn718	RE24O3	RE23O3	RE22O3	RE21O3	RE20O3	RE19O3	RE18O3	RE17O3
Fn719	RE32O3	RE31O3	RE30O3	RE29O3	RE28O3	RE27O3	RE26O3	RE25O3
Fn720	RE08O4	RE07O4	RE06O4	RE05O4	RE04O4	RE03O4	RE02O4	RE01O4
Fn721	RE16O4	RE15O4	RE14O4	RE13O4	RE12O4	RE11O4	RE10O4	RE09O4
Fn722	RE24O4	RE23O4	RE22O4	RE21O4	RE20O4	RE19O4	RE18O4	RE17O4
Fn723	RE32O4	RE31O4	RE30O4	RE29O4	RE28O4	RE27O4	RE26O4	RE25O4

Note

NOTE

When bit 0(SSE) of parameter No.3791 is set to 1, following signals are not effective.

- Spindle motor speed command signals	R01I to R12I, R01I2 to R12I2, R01I3 to R12I3, R01I4 to R12I4
- S12-bit code signals	R01O to R12O, R01O2 to R12O2, R01O3 to R12O3, R01O4 to R12O4
- Actual spindle speed signals	AR00 to AR15, AR002 to AR152, AR003 to AR153, AR004 to AR154

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3791								SSE

[Input type] Parameter input
[Data type] Bit spindle

#0 SSE The resolution enabled for the spindle speed command:
0: depend to the bit 4(SSI) of parameter No.3798.
1: is a maximum spindle speed/1048575 [min^{-1}].

> **NOTE**
> 1 When this parameter is set, the power must be turned off before operation is continued.
> 2 When bit 0 (SSE) of parameter No.3791 is set to 1, bit 4 (SSI) of parameter No.3798 is not effective.

	#7	#6	#5	#4	#3	#2	#1	#0
3798				SSI	SDP			

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#3 SDP High-precision spindle speed control is:
0: Not used.
1: Used.

#4 SSI The resolution enabled for the spindle speed command is:
0: Maximum spindle speed/4095 [min^{-1}].
1: Maximum spindle speed/16383 [min^{-1}].

	#7	#6	#5	#4	#3	#2	#1	#0
3709					MRS	MSI		

[Input type] Parameter input
[Data type] Bit path

#2 MSI In multi-spindle control, the Spindle motor speed command selection signal SINDs signal is valid
0: Only when the first spindle is valid (Spindle motor speed command selection signal SINDs for the 2nd, 3rd spindle becomes ineffective) (TYPE-A)
1: For each spindle irrespective of whether the spindle is selected (Each spindle has its own Spindle motor speed command selection signal SINDs). (TYPE-B)

#3 MRS When the actual spindle speed signals and S 12-bit code signals are output in multi-spindle control:
0: The signals common to the first spindle and second spindle are used, and the signals for the spindle selected by the spindle selection signal are output.
1: The signals for the first spindle and the signals for the second spindle are output separately.

Alarm and message

Number	Message	Description

SP1257	ILLEGAL PARAMETER (No.3791#0)	Bit 0(SSE) of parameter No.3791 is illegal setting.
		Causes of the alarm are as follows.
		- The setting of bit 0 (SSE) of parameter No.3791 is different because of the master and slave. Please change the following function to the same setting.
		  - Spindle synchronous control
		  - Spindle command synchronous control
		  - Polygon turning with two spindles
		- The bit 0 (SSE) of parameter No.3791 is set to 1 the spindle control software that does not support this function. Please set 0 to bit 0 (SSE) of parameter No.3791.

Diagnosis data

- When bit 0 (SSE) of No.3791 is set to 1, the spindle motor speed is displayed to the diagnosis data No.1547. And then, the diagnosis No.411 is displayed 0.

1547	Spindle motor speed data (2 word)

[Data type] 2 word spindle
[Unit] min^{-1}

- When this function is effective, bit 0 of diagnosis data No.1570 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1570								S2W

S2W A spindle speed command is the resolution of maximum spindle speed/1048575 [min^{-1}].

10.28 SPINDLE CONTROL MODE CHANGING BY PROGRAM COMMAND

Overview

Spindle control mode can be changed by commanding M code which is set to parameter. The following spindle control modes can be changed.

- Spindle orientation
- Cs contour control

Programming PMC ladder for changing the spindle mode is unnecessary.

Explanation

- Spindle orientation

Set M code for spindle orientation ON to parameter No.25800, and set M code for spindle orientation OFF to parameter No.25801. If spindle orientation by program command is not available, set 0 to parameters No.25800, and No.25801 of the spindle.
If spindle selection by address P of Multi-spindle control is enabled (bit 3 (MPP) of parameter No.3703 is 1), spindle executing spindle orientation can be selected by address P. Spindles executing spindle orientation by program command use the same M code for spindle orientation ON, and the same M code

for spindle orientation OFF. Set the same M code to parameter of spindle executing spindle orientation (refer to Example (2)).

When M code for spindle orientation ON or M code for spindle orientation OFF is commanded, auxiliary function code signals M00 to M31<Fn010 to Fn013> and auxiliary function strobe signal MF<Fn007.0> are not output.

DI signals for spindle orientation are available even if M code for spindle orientation ON and M code for spindle orientation OFF are available. However, DI signals are not available from commanding M code for spindle orientation ON to canceling spindle orientation. If M code for spindle orientation ON or M code for spindle orientation OFF is commanded when spindle orientation command signal ORCMA<Gn070.6> (first spindle), ORCMB<Gn074.6> (second spindle), ORCMC<Gn204.6> (third spindle), ORCMD<Gn266.6> (fourth spindle) is "1", alarm PS0553, "SP-MODE CHANGE IMPOSSIBLE" is issued.

There are two methods for specifying stop position of spindle orientation.

- Orientation with the fixed stop position (bit 0 (ORT) of parameter No.3729 is 0)
  Stop position of spindle orientation can be set to parameters No.4031 (MAIN spindle) and No.4204 (SUB spindle).
- Orientation with the stop position set externally (bit 0 (ORT) of parameter No.3729 is 1)
  Stop position of spindle orientation can be commanded by address R. Increment system of address R is that of reference axis (parameter No.1031). Least input increment for stop position of spindle orientation is 360°/4096=0.088° when bit 0 (ORPUNT) of parameter No.4542 is 0 or 360°/32768=0.011° when ORPUNT is 1. If address R is not commanded, stop position of spindle orientation is 0°. If M code for spindle orientation ON is commanded again after commanding M code for spindle orientation ON, spindle is moved to the stop position newly commanded. Spindle took a shortcut and move angle is within ±180° at this time.

Format

M_ (P_) (R_) ; Spindle orientation ON

M : M code for spindle orientation ON (parameter No.25800)
P : Spindle selection of Multi-spindle control
R : Stop position of spindle orientation (when bit 0 (ORT) of parameter No.3729 is 1)

M_ (P_) ; Spindle orientation OFF

M : M code for spindle orientation OFF (parameter No.25801)
P : Spindle selection of Multi-spindle control

Example (1) Spindle orientation of first spindle and second spindle by program command

Table 10.28 (a) Example of parameter setting

	Parameter		
	No.25800	**No.25801**	**No.3729#0**
First spindle	19	18	1
Second spindle	119	118	1
Third spindle	0	0	0

Example of machining program

Program	Description
O1001 ;	
:	
N10 M03 S100 ;	First spindle rotates at a speed of 100[min^{-1}].
:	
N20 M19 R90.0 ;	Spindle orientation of first spindle is executed. Stop position of the spindle is 90°.
:	
:	
N30 M19 R180.0 ;	Spindle orientation of first spindle is executed. The spindle moves from 90° to 180°.
:	
:	
N40 M18 ;	Spindle orientation of first spindle is canceled. First spindle does not rotate until S code is newly commanded.
:	
:	
:	
N50 M119 ;	Spindle orientation of second spindle is executed. Stop position of the spindle is 0°.
:	
:	
N60 M118 ;	Spindle orientation of second spindle is canceled. Second spindle does not rotate until S code is newly commanded.
:	
:	
:	
N70 M03 S200 ;	First spindle rotates at a speed of 200[min^{-1}].
:	
:	

Example (2) Spindle selection by address P of Multi-spindle control

Table 10.28 (b) Example of parameter setting

	Parameter				
	No.25800	**No.25801**	**No.3781**	**No.3729#0**	**No.3703#3**
First spindle	19	18	1	1	1
Second spindle	19	18	2	1	
Third spindle	0	0	3	0	

Example of machining program

O1002 ;	
:	
N10 M03 S100 P1 ;	First spindle rotates at a speed of 100[min^{-1}].
:	
N20 M19 R90.0 P1 ;	Spindle orientation of first spindle is executed. Stop position of the spindle is 90°.
:	
:	
N30 M19 R180.0 P1 ;	Spindle orientation of first spindle is executed. The spindle moves from 90° to 180°.
:	
:	
N40 M18 P1 ;	Spindle orientation of first spindle is canceled. First spindle does not rotate until S code is newly commanded.
:	
:	
:	
N50 M19 P2 ;	Spindle orientation of second spindle is executed. Stop position of the spindle is 0°.
:	
:	
N60 M18 P2 ;	Spindle orientation of second spindle is canceled. Second spindle does not rotate until S code is newly commanded.
:	
:	
:	
N70 M03 S200 P1 ;	First spindle rotates at a speed of 200[min^{-1}].
:	
N80 M19 P3 ;	Alarm PS0552, "SP-MODE CHANGE COMMAND ERROR" is issued because parameter No.25800 of third spindle is 0.
:	
:	
:	

If M code for spindle orientation ON is commanded, spindle speed command is set to 0 in order not to rotate the spindle when spindle orientation is canceled. Therefore, S code has to be newly commanded for rotating the spindle after spindle orientation OFF. In Multi-spindle control, if spindle selection method by spindle selection signals SWS1 to SWS4 <Gn027.0, Gn027.1, Gn027.2>, <Gn026.3> is used, select spindle by spindle selection signals before commanding S code. If S code has not been commanded in the path from which M code for spindle orientation ON is commanded and only M03/M04 is commanded, alarm PS0554, "S-CODE IS NOT COMMANDED" is issued in order not to start machining without spindle rotation. If only M code except for M03/M04 is commanded, alarm PS0554 is not issued. If M code except for M03/M04 is used for spindle rotation, be sure to command S code in order not to start machining without spindle rotation.

Example (3) example of alarm PS0554 by only M03/M04 command

Example of machining program

O1003 ;	
:	
N10 M03 S100 P1 ;	First spindle rotates at a speed of 100[min^{-1}].
:	
N20 M19 P1 ;	Spindle orientation of first spindle is executed.
:	
N30 M18 P1 ;	Spindle orientation of first spindle is canceled. First spindle does not rotate until S code is newly commanded.
:	
:	
:	
N40 M03 ;	Alarm PS0554, "S-CODE IS NOT COMMANDED" is issued because S code has not been commanded.
:	
:	
:	

NOTE

1 G code using address P, address Q, or address R and M code for spindle orientation ON or M code for spindle orientation OFF cannot be commanded in the same block. When spindle selection by address P is available, if machine coordinate system select command (G53) is commanded, alarm PS0552, "SP-MODE CHANGE COMMAND ERROR" is issued.
When spindle selection by address P is available and the addition of workpiece coordinate system 48 pairs or 300 pairs is available, if selecting a workpiece coordinate system (G54) is commanded, alarm PS0552, “SP-MODE CHANGE COMMAND ERROR” is issued.

2 Spindle orientation by program command is not canceled by emergency stop, reset. However, if emergency stop or reset is executed during M code for spindle orientation ON, spindle orientation is canceled.

3 S code has to be newly commanded for rotating the spindle after canceling spindle orientation by program command. If S code has not been commanded in the path from which M code for spindle orientation ON is commanded and only M03/M04 is commanded, alarm PS0554, "S-CODE IS NOT COMMANDED" is issued. If only M code except for M03/M04 is commanded, alarm PS0554 is not issued. Be sure to command S code if M code except for M03/M04 is used for spindle rotation in order not to start machining without spindle rotation.

4 Multiple M commands in a single block cannot be used. Don’t command other M codes with M code for spindle orientation ON or M code for spindle orientation OFF in the same block.

5 Spindle orientation by program command cannot be used if spindle orientation by a magnetic sensor is used.

6 Enabling spindle serial output (bit 5 (SSN) of parameter No.8133 is 0) and spindle orientation (bit 4 (NOR) of parameter No.8135 is 0) is required in order to use this function.

- Cs contour control

Set M code for Cs contour control ON to parameter No.25802, and set M code for Cs contour control OFF to parameter No.25803. If Cs contour control by program command is not available, set 0 to parameters No.25802 and No.25803 of the spindle.
If spindle selection by address P of Multi-spindle control is enabled (bit 3 (MPP) of parameter No.3703 is 1), spindle changing Cs contour control mode can be selected by address P. Spindles changing Cs contour control mode by program command use the same M code for Cs contour control ON, and the same M code for Cs contour control OFF. Set the same M code to parameter of spindle changing Cs contour control mode (refer to Example (5)).

When M code for Cs contour control ON or M code for Cs contour control OFF is commanded, auxiliary function code signals M00 to M31<Fn010 to Fn013> and auxiliary function strobe signal MF<Fn007.0> are not output.

Cs contour control axis coordinate establishment is available when bit 2 (CSF) of parameter No.3712 is 1 and bit 5 (CSPTRE) of parameter No.4353 is 1. If "Q1" is commanded with M code for Cs contour control ON in the same block, coordinate of Cs contour control axis is established after changing Cs contour control.

Cs contour control change signal CON<Gn027.7>, Cs contour control change signals in each axis CONS1 to CONS4<Gn274.0 to Gn274.3>, or Cs axis coordinate establishment request signals CSFI1 to CSFI4<Gn274.4 to Gn274.7> are available even if M code for Cs contour control ON or M code for Cs contour control OFF are available. However, DI signals are not available from commanding M code for Cs contour control ON to canceling Cs contour control. If M code for Cs contour control ON or M code for Cs contour control OFF is commanded when Cs contour control change signal or Cs contour control change signal in each axis is "1", alarm PS0553, "SP-MODE CHANGE IMPOSSIBLE" is issued.

Format

M_ (P_) (Q1) ; Cs contour control ON

M : M code for Cs contour control ON (parameter No.25802)
P : Spindle selection of Multi-spindle control
Q1 : Cs contour control axis coordinate establishment

M_ (P_) ; Cs contour control OFF

M : M code for Cs contour control OFF (parameter No.25803)
P : Spindle selection of Multi-spindle control

Example (4) Cs contour control of first spindle and second spindle by program command

Table 10.28 (c) Example of parameter setting

	Parameter			
	No.25802	No.25803	No.4353#5	No.3712#2
First spindle	50	51	1	
Second spindle	150	151	1	1
Third spindle	0	0	0	

Example of machining program

```
O1004 ;
   :
N10 M50 ;          First spindle changes to Cs contour control.
   :
N20 M51 ;          First spindle changes to spindle rotation control.
   :
N30 M150 Q1 ;      Second spindle changes to Cs contour control.
   :               Cs contour control axis coordinate establishment is
   :               executed.
   :
N40 M151 ;         Second spindle changes to spindle rotation control.
   :
   :
```

Example (5) Spindle selection by address P of Multi-spindle control

Table 10.28 (d) Example of parameter setting

<table>
<tr><th rowspan="2"></th><th colspan="6">Parameter</th></tr>
<tr><th>No.25802</th><th>No.25803</th><th>No.3781</th><th>No.4353#5</th><th>No.3712#2</th><th>No.3703#3</th></tr>
<tr><td>First spindle</td><td>50</td><td>51</td><td>1</td><td>1</td><td rowspan="3">1</td><td rowspan="3">1</td></tr>
<tr><td>Second spindle</td><td>50</td><td>51</td><td>2</td><td>1</td></tr>
<tr><td>Third spindle</td><td>0</td><td>0</td><td>3</td><td>0</td></tr>
</table>

Example of machining program

```
O1005 ;
   :
N10 M50 P1 ;       First spindle changes to Cs contour control.
   :
N20 M51 P1 ;       First spindle changes to spindle rotation control.
   :
N30 M50 P2 Q1 ;    Second spindle changes to Cs contour control.
   :               Cs contour control axis coordinate establishment is
   :               executed.
   :
N40 M51 P2 ;       Second spindle changes to spindle rotation control.
   :
N50 M50 P3 ;       Alarm PS0552, "SP-MODE CHANGE COMMAND
   :               ERROR" is issued because parameter No.25802 of
   :               third spindle is 0.
   :
```

If M code for Cs contour control ON is commanded, spindle speed command is set to 0 in order not to rotate the spindle when Cs contour control is canceled. Therefore, S code has to be newly commanded for rotating the spindle after Cs contour control OFF. In Multi-spindle control, if spindle selection method by spindle selection signals SWS1 to SWS4 <Gn027.0, Gn027.1, Gn027.2>, <Gn026.3> is used, select spindle by spindle selection signals before commanding S code. If S code has not been commanded in the path from which M code for Cs contour control ON is commanded and only M03/M04 is commanded, alarm PS0554, "S-CODE IS NOT COMMANDED" is issued in order not to start machining without spindle rotation. If only M code except for M03/M04 is commanded, alarm PS0554 is not issued. If M code except for M03/M04 is used for spindle rotation, be sure to command S code in order not to start machining without spindle rotation.

Example (6) example of alarm PS0554 by only M03/M04 command

Example of machining program

Program	Description
O1006 ;	
:	
N10 M05 ;	First spindle stops.
:	
N20 M50 P1 ;	First spindle changes to Cs contour control.
:	
N30 M51 P1 ;	First spindle changes to spindle rotation control.
:	
N40 M03 ;	Alarm PS0554, "S-CODE IS NOT COMMANDED" is issued because S code has not been commanded.
:	
:	
:	

If M code for spindle orientation ON is commanded after Cs contour control ON by program command, Cs contour control is canceled, and spindle orientation is executed.

Example (7)
M code command for spindle orientation ON after Cs contour control ON by program command

Example of machining program

Program	Description
O1007 ;	
:	
N10 M50 P1 ;	First spindle changes to Cs contour control.
:	
N20 M19 R270.0 P1 ;	First spindle changes to spindle rotation control, then executes spindle orientation. Stop position of the spindle is 270°.
:	
:	
:	

Similarly, if M code for Cs contour control ON is commanded during spindle orientation ON by program command, spindle orientation is canceled, and the spindle changes to Cs contour control.

Example (8)
M code command for Cs contour control ON during spindle orientation ON by program command

Example of machining program

Program	Description
O1008 ;	
:	
N10 M19 P1 ;	Spindle orientation of first spindle is executed. Stop position of the spindle is 0°.
:	
:	
N20 M50 P1 ;	First spindle cancels spindle orientation, then changes Cs contour control.
:	
:	
:	

NOTE

1 G code using address P, address Q, or address R and M code for Cs contour control ON or M code for Cs contour control OFF cannot be commanded in the same block. When spindle selection by address P is available, if machine coordinate system select command (G53) is commanded, alarm PS0552, "SP-MODE CHANGE COMMAND ERROR" is issued.
When spindle selection by address P is available and the addition of workpiece coordinate system 48 pairs or 300 pairs is available, if selecting a workpiece coordinate system (G54) is commanded, alarm PS0552, "SP-MODE CHANGE COMMAND ERROR" is issued.

2 Cs contour control by program command is not canceled by emergency stop, reset. However, if emergency stop or reset is executed during M code for Cs contour control ON, Cs contour control is canceled.

3 In changing Cs contour control, if CW command signals SFR or CCW command signals SRV has to be changed by PMC ladder, Cs contour control change completion signal FSCSL<Fn044.1> or Cs contour control change completion signal in each axis FCSS1 to FCSS4<Fn274.0 to Fn274.4> is available (bit 3 (NCS) of parameter No.3729 has to be set to 1 (switching to Cs contour control is completed even when the spindle activating current is off)).

4 S code has to be newly commanded for rotating the spindle after canceling Cs contour control by program command. If S code has not been commanded in the path from which M code for Cs contour control ON is commanded and only M03/M04 is commanded, alarm PS0554, "S-CODE IS NOT COMMANDED" is issued. If only M code except for M03/M04 is commanded, alarm PS0554 is not issued. Be sure to command S code if M code except for M03/M04 is used for spindle rotation in order not to start machining without spindle rotation.

5 Multiple M commands in a single block cannot be used. Don't command other M codes with M code for Cs contour control ON or M code for Cs contour control OFF in the same block.

6 Enabling spindle serial output (bit 5 (SSN) of parameter No.8133 is 0) and Cs contour control (bit 2 (SCS) of parameter No.8133 is 1) is required in order to use this function.

- Spindle control mode OFF

Spindle orientation by program command or Cs contour control by program command can be canceled by M codes which are set to parameter No.25804, No.25805, and No.25806. The spindle whose parameters (No.25804, No.25805, and No.25806) are set to 0 cannot use M codes for spindle control mode OFF.

If spindle selection by address P of Multi-spindle control is enabled (bit 3 (MPP) of parameter No.3703 is 1), spindle canceling spindle control mode can be selected by address P. Spindles canceling spindle control mode by program command use the same M code for spindle control mode OFF. Set the same M code to parameter of spindle canceling spindle control mode (refer to Example (10)).

When M code for spindle control mode OFF (parameter No.25804, No.25805, and No.25806) is commanded, auxiliary function code signals M00 to M31<Fn010 to Fn013> and auxiliary function strobe signal MF<Fn007.0> are output.

Format

M_ (P_) ;   Spindle control mode OFF

M : M code for spindle control mode OFF (parameter No.25804, No.25805, and No.25806)

P : Spindle selection of Multi-spindle control

Example (9) Spindle rotation after spindle control mode OFF

Table 10.28 (e) Example of parameter setting

	Parameter		
	No.25804	**No.25805**	**No.25806**
First spindle	3	4	5
Second spindle	103	104	105
Third spindle	0	0	0

Example of machining program

Program	
O1009 ;	
:	
N10 M19 ;	Spindle orientation of first spindle is executed.
:	
N20 M03 S100 ;	Spindle orientation of first spindle is canceled. First spindle rotates at a speed of 100[min^{-1}].
:	
:	
N30 M150 ;	Second spindle changes to Cs contour control.
:	
:	
N40 M104 S200 ;	Second spindle changes to spindle rotation control. Second spindle rotates at a speed of 200[min^{-1}].
:	
:	
:	

Example (10) Spindle selection by address P of Multi-spindle control

Table 10.28 (f) Example of parameter setting

	Parameter				
	No.25804	**No.25805**	**No.25806**	**No.3781**	**No.3703#3**
First spindle	3	4	5	1	
Second spindle	3	4	5	2	1
Third spindle	0	0	0	3	

Example of machining program

Program	Description
O1010 ;	
:	
N10 M19 P1 ;	Spindle orientation of first spindle is executed.
:	
N20 M03 S100 P1 ;	Spindle orientation of first spindle is canceled. Spindle rotates at a speed of 100[min^{-1}].
:	
:	
N30 M50 P2 ;	Second spindle changes to Cs contour control.
:	
:	
N40 M04 S200 P2 ;	Second spindle changes to spindle rotation control. Second spindle rotates at a speed of 200[min^{-1}].
:	
:	
:	

NOTE

1 G code using address P, address Q, or address R and M code for spindle control mode OFF cannot be commanded in the same block. When spindle selection by address P is available, if machine coordinate system select command (G53) is commanded, alarm PS0552, "SP-MODE CHANGE COMMAND ERROR" is issued.
When spindle selection by address P is available and the addition of workpiece coordinate system 48 pairs or 300 pairs is available, if selecting a workpiece coordinate system (G54) is commanded, alarm PS0552, "SP-MODE CHANGE COMMAND ERROR" is issued.

2 S code has to be newly commanded for rotating the spindle after canceling spindle control mode by M code. If S code has not been commanded and only M03/M04 is commanded, alarm PS0554, "S-CODE IS NOT COMMANDED" is issued. If only M code except for M03/M04 is commanded, alarm PS0554 is not issued. Be sure to command S code if M code except for M03/M04 is used for spindle rotation in order not to start machining without spindle rotation.

3 If M03/M04 is set as M code for spindle control mode OFF, S code has to be command with M03/M04 in the same block. If S code is not commanded, alarm PS0554 is issued.

4 If Multiple M commands in a single block is used, command M code for spindle control mode OFF as the first M code.

- Spindle control mode OFF by signal

Spindle orientation by program command or Cs contour control by program command can be canceled by spindle control mode off signals MDOFF1 to MDOFF4<Gn586.4 to Gn586.7>. If you want to cancel spindle orientation or Cs contour control by emergency stop, reset, and so on, spindle control mode can be canceled by spindle control mode off signals MDOFF1 to MDOFF4<Gn586.4 to Gn586.7>. If spindle control mode off signal MDOFF1 to MDOFF4<Gn586.4 to Gn586.7> is set to "1", spindle control mode is canceled. If spindle orientation is canceled, spindle orientation completion signal ORARA to ORARD<Fn045.7, Fn049.7, Fn168.7, Fn266.7> is set to "0". Then, set "0" to spindle control mode off signals MDOFF1 to MDOFF4<Gn586.4 to Gn586.7>. If Cs contour control is canceled, Cs contour control change completion signal FSCSL<Fn044.1> or Cs contour control change completion signal in each axis FCSS1 to FCSS4<Fn274.0 to Fn274.3> is set to "0". Then, set "0" to spindle control mode off signals MDOFF1 to MDOFF4<Gn586.4 to Gn586.7>.

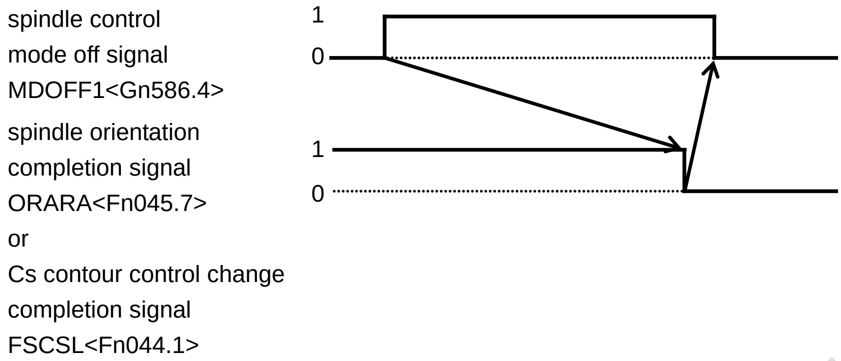


Fig. 10.28 (a) Spindle control mode OFF by signal

Spindle orientation by orientation command signal ORCMA to ORCMD<Gn070.6, Gn074.6, Gn204.6, Gn266.6> cannot be canceled by spindle mode off signals MDOFF1 to MDOFF4<Gn586.4 to Gn586.7>. Similarly, Cs contour control by Cs contour control change signal CON<Gn027.7> or Cs contour control change signals in each axis CONS1 to CONS4<Gn274.0 to Gn274.3> cannot be canceled by spindle mode off signals MDOFF1 to MDOFF4<Gn586.4 to Gn586.7>.
When spindle control mode off signal MDOFF1 to MDOFF4<Gn586.4 to Gn586.7> is "1", if M code for spindle orientation ON set to parameter No.25800 or M code for Cs contour control ON set to parameter No.25802 is commanded, alarm PS0553, "SP-MODE CHANGE IMPOSSIBLE" is issued.

- Multi-spindle address P signals

If bit 0 (ADP) of parameter No.25807 is set to 1, the value of address P which is commanded to select mode changing spindle is output to multi-spindle address P signals MSP00 to MSP15<Fn160.0 to Fn161.7>.

Limitation

- Command with a signal

M code for spindle control mode changing is not output to MDI program by auxiliary function output in program restart function.

- Auxiliary function output in moving axis

If M code for spindle control mode changing is commanded in G50.9 block, alarm PS5330 "G50.9 FORMAT ERROR" is issued.

- Hypothetical Cs axis

Spindle control mode changing by program command is not available for hypothetical Cs axis. Set 0 to parameters Nos.25800 to 25806.

- Spindle control with servo motor

Spindle control mode changing by program command is not available for spindle controlled with servo motor. Set 0 to parameter Nos.25800 to 25806.

M

- Canned cycle for drilling

In Fine boring cycle (G76) and Back boring cycle (G87), M19 is output in order to execute spindle orientation during canned cycle. If M19 is set as M code for spindle orientation by program command, M19 in Fine boring cycle (G76) and Back boring cycle (G87) does not execute spindle orientation by program command, auxiliary function code signals M00 to M31<Fn010 to Fn013> and auxiliary function strobe signal MF<Fn007.0> are output. Spindle orientation M19 in Fine boring cycle (G76) and Back boring cycle (G87) has to execute by PMC ladder.

Signal

Spindle control mode off signals MDOFF1 to MDOFF4 <Gn586.4 to Gn586.7>

[Classification] Input signal

[Function] In spindle control mode changing by program command, spindle orientation by program command or Cs contour control by program command is canceled.

[Operation] If spindle control mode off signal MDOFF1 to MDOFF4<Gn586.4 to Gn586.7> is set to "1", spindle orientation by program command or Cs contour control by program command is canceled. If spindle orientation completion signal ORARA to ORARD<Fn045.7, Fn049.7, Fn168.7, Fn266.7>, or Cs contour control change completion signal FSCSL<Fn044.1> (Cs contour control change completion signal in each axis FCSS1 to FCSS4<Fn274.0 to Fn274.3>) is set to "0", set "0" to spindle control mode off signals MDOFF1 to MDOFF4<Gn586.4 to Gn586.7>.

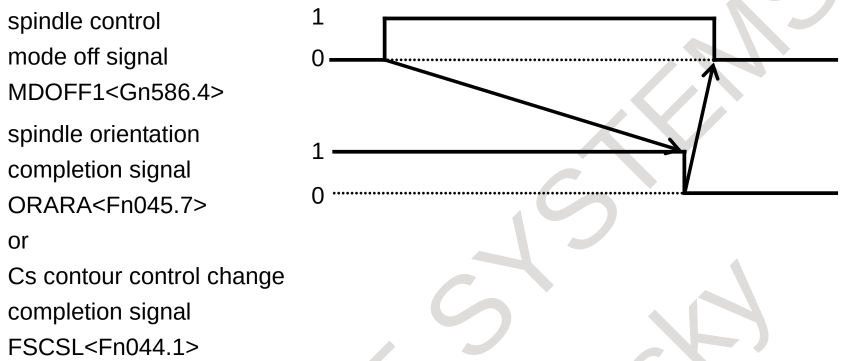


Fig. 10.28 (b) Spindle control mode OFF by signal

Multi-spindle address P signals MSP00 to MSP15 <Fn160.0 to Fn161.7>

[Classification] Output signal

[Function] The P value specified last by the S_P_; command is output.
When extended spindle names are used, the value in parameter No. 3781 corresponding to the spindle number of the spindle name specified last is output.

[Output cond.] When multi-spindle control by using address P is enabled (bit 3 (MPP) of parameter No. 3703 is 1), the P value specified in the S_P_; command is output. When no S_P_; command has been specified even once since power-up, the initial P value set in parameter No. 3775 is output.
When extended spindle names are used, the value in parameter No. 3781 corresponding to the spindle number of the spindle name specified last is output. When no spindle name has been specified even once, the value set in parameter No. 3775 is output.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn586	MDOFF4	MDOFF3	MDOFF2	MDOFF1				
Fn160	MSP07	MSP06	MSP05	MSP04	MSP03	MSP02	MSP01	MSP00
Fn161	MSP15	MSP14	MSP13	MSP12	MSP11	MSP10	MSP09	MSP08

Parameter

25800	M code for spindle orientation ON

> **NOTE**
> 1 When this parameter is set, the power must be turned off before operation is continued.
> 2 Don’t set M code used for any other function.
> 3 The M code set in this parameter prevents buffering.
> 4 Auxiliary function code signals M00 to M31<Fn010 to Fn013> and auxiliary function strobe signal MF<Fn007.0> are not output.
> 5 If this parameter is set to 0, the spindle cannot use spindle orientation by program command.

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 0 to 99999999

Set M code for spindle orientation ON.

25801	M code for spindle orientation OFF

> **NOTE**
> 1 When this parameter is set, the power must be turned off before operation is continued.
> 2 Don’t set M code used for any other function.
> 3 Auxiliary function code signals M00 to M31<Fn010 to Fn013> and auxiliary function strobe signal MF<Fn007.0> are not output.
> 4 If this parameter is set to 0, the spindle cannot use spindle orientation by program command.

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 0 to 99999999

Set M code for spindle orientation OFF.

25802	M code for Cs contour control ON

> **NOTE**
> 1 When this parameter is set, the power must be turned off before operation is continued.
> 2 Don’t set M code used for any other function.
> 3 The M code set in this parameter prevents buffering.
> 4 Auxiliary function code signals M00 to M31<Fn010 to Fn013> and auxiliary function strobe signal MF<Fn007.0> are not output.
> 5 If this parameter is set to 0, the spindle cannot use Cs contour control by program command.

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 0 to 99999999

Set M code for Cs contour control ON.

25803	M code for Cs contour control OFF

NOTE
1 When this parameter is set, the power must be turned off before operation is continued.
2 Don't set M code used for any other function.
3 Auxiliary function code signals M00 to M31<Fn010 to Fn013> and auxiliary function strobe signal MF<Fn007.0> are not output.
4 If this parameter is set to 0, the spindle cannot use Cs contour control by program command.

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 0 to 99999999
Set M code for Cs contour control OFF.

25804	M code for spindle control mode OFF 1
25805	M code for spindle control mode OFF 2
25806	M code for spindle control mode OFF 3

NOTE
1 When this parameter is set, the power must be turned off before operation is continued.
2 Don't set M code used for any other function.
3 If this parameter is set to 0, the spindle cannot use M code for spindle control mode OFF.

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 0 to 99999999
Set M code for spindle control mode OFF.

	#7	#6	#5	#4	#3	#2	#1	#0
25807								ADP

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 ADP In Spindle control mode changing by program command, when spindle selection by address P of Multi-spindle control is enabled, multi-spindle address P signals MSP00 to MSP15<Fn160.0 to Fn161.7> (the value is address P commanded with M code for spindle control mode changing) is:
0: Not output.
1: Output.

	#7	#6	#5	#4	#3	#2	#1	#0
3703					MPP			

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#3 MPP In multi-spindle control, a spindle selection using a programmed command instead of using the signals (SWS1 to SWS4<Gn027.0 to Gn027.2, Gn026.3>) is:
0: Not made.
1: Made.

> **NOTE**
> When this parameter is set to 1, set parameter No.3781 at the same time.

	#7	#6	#5	#4	#3	#2	#1	#0
3712						CSF		

[Input type] Parameter input
[Data type] Bit

#2 CSF In the Cs contour control mode, the function for setting machine coordinates and absolute coordinates based on the machine position of the spindle if the origin is already set up is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3729					NCSs			ORTs

[Input type] Parameter input
[Data type] Bit spindle

#0 ORTs When a serial spindle is used, the spindle orientation function of stop position external setting type based on the position coder is:
0: Not performed.
1: Performed.

> **NOTE**
> When bit 0 (ORPUNT) of No.4542 is set to 1, if bit 0 (ORTs) of parameter No.3729 is changed, the power must be turned off before operation is continued.

#3 NCSs When the Cs contour control mode is set:
0: Switching to Cs contour control is completed when the spindle activating current is on (the spindle amplifier is ready for operation in the Cs contour control mode).
1: Switching to Cs contour control is completed even when the spindle activating current is off (the spindle amplifier is not ready for operation in the Cs contour control mode).

If this parameter is set to 1, the Cs contour control switch end signal is output without waiting for the spindle to decelerate to a stop.

3781	P code for selecting the spindle in multi-spindle control

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767

If bit 3 (MPP) of parameter No. 3703 is set to 1, set the P code to select each spindle under multi-spindle control. Specify the P code in a block containing the S command.

[Example] If the P code value for selecting the second spindle is set to 2,
S1000 P2;
Causes the second spindle to rotate at S1000.

NOTE

1. This parameter is valid if bit 3 (MPP) of parameter No. 3703 is set to 1.
2. If this parameter is set to 0, the corresponding spindle cannot be selected by a P code.
3. Under multi-path control, the P code specified here is valid for each path.
   For instance, if the P code to select the first spindle of path 2 is set to 21, specifying S1000 P21; in path 1 causes the first spindle of path 2 to be rotated at S1000.
4. Identical P code values cannot be used for different spindles. (Identical P code values cannot be used even if the paths are different.)
5. When this parameter is used (when bit 3 (MPP) of parameter No. 3703 is set to 1), the spindle command selection signal is invalid.
6. To use this parameter, enabling the multi-spindle control (bit 3 (MSP) of parameter No.8133 is 1) is needed.

4031	Stop position in orientation by a position coder (MAIN spindle)

4204	Stop position in orientation by a position coder (SUB spindle)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] Depend on the bit 0 (ORPUNT) of No.4542 setting (Refer to the table below)

Bit 0 (ORPUNT) of No.4542	Least input increment	Valid data range
0	360/4096= 0.08789	0 to 4095
1	360/32768= 0.01098	0 to 32767

Each of these parameters specifies a stop position in orientation by a position coder.
These parameters are valid when bit 0 (ORT) of parameter No.3729 is set to 0.

4075	Orientation completion signal detection level (limits of in-position)（MAIN spindle）

4226	Orientation completion signal detection level (limits of in-position)（SUB spindle）

[Input type] Parameter input

[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] Depend on the bit 0 (ORPUNT) of No.4542 setting (Refer to the table below)

Bit 0 (ORPUNT) of No.4542	Least input increment	Valid data range
0	360/4096= 0.08789	0 to 100
1	360/32768= 0.01098	0 to 1000

This data is used to set the detecting level of orientation completion signal (ORARA). ORARA is assumed to be 1 if the position error is within the setting.

4077	Stop position shift amount in orientation by a position coder(MAIN spindle)

4228	Stop position shift amount in orientation by a position coder(SUB spindle)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] Depend on the bit 0 (ORPUNT) of No.4542 setting (Refer to the table below)

Bit 0 (ORPUNT) of No.4542	Least input increment	Valid data range
0	360/4096= 0.08789	-4095 to 4095
1	360/32768= 0.01098	-32767 to 32767

Each of these parameters specifies a stop position shift amount in orientation by a position coder.
This parameter is valid no matter what the setting of bit 0 (ORT) of parameter No.3729 is.

	#7	#6	#5	#4	#3	#2	#1	#0
4353			CSPTRE					

[Input type] Parameter input
[Data type] Bit spindle

#5 CSPTRE Specifies whether to enable Cs axis positional data transfer, as follows:
0: Disable.
1: Enable.

To use this function, reset bit 7 (RFCHK3) of parameter No. 4016 to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
4542								ORPUNT

[Input type] Parameter input
[Data type] Bit spindle

#0 ORPUNT Least input increment for Stop position of spindle orientation, Stop position shift amount, In-position width
0: 360/4096 = 0.08789 [deg]
1: 360/32768 = 0.01098 [deg]

⚠ **CAUTION**

In the αi position coder, the effect is not achieved by the fine resolution application of stop position because the sensor resolution is 4096p/rev.

NOTE

To use this function, the corresponding serial spindle software is required.

Alarm and message

Number	Message	Description
PS0552	SP-MODE CHANGE COMMAND ERROR	Spindle control mode changing command is illegal. Causes of the alarm are as follows. - Spindle control mode changing is commanded for disabling spindle. - Multiple M commands are commanded in spindle orientation ON/OFF command or Cs contour control ON/OFF command. - Illegal G code (G53, G54) is commanded at spindle control mode changing block.
PS0553	SP-MODE CHANGE IMPOSSIBLE	Spindle control mode changing command cannot execute. Causes of the alarm are as follows. - Spindle control mode changing by program command is commanded during spindle orientation or Cs contour control by DI signal. - During spindle control mode changing, spindle control mode changing is commanded from the other path. - Spindle control mode changing is commanded when spindle control mode off signal MDOFF1 to MDOFF4<Gn586.4 to Gn586.7> is "1".
PS0554	S-CODE IS NOT COMMANDED	M03/M04 is commanded alone without S code.
PW0053	SP-MODE CHANGE SETTING ERROR	The setting of parameters Nos.25800 to 25806 is illegal. Causes of the alarm are as follows. - The setting value is out of range. - The same value is set when spindle selection by address P is not used. - The same value is not set to all spindles when spindle selection by address P is used. - The value except for 0 is set when spindle orientation or Cs contour control is not enabled (bit 4 (NOR) of parameter No.8135 is 1 or bit 2 (SCS) of parameter No.8133 is 0). - One of M code of spindle orientation ON/OFF is set. - One of M code of Cs contour control ON/OFF is set.

11 TOOL FUNCTIONS

11.1 TOOL FUNCTIONS OF LATHE SYSTEM

T

Overview

When address T followed by a numeric value is specified, the code signal and strobe signal are sent to the machine and are used to select a tool on the machine side. Just one T code can be specified in a block.

When a move command and T code are specified within the same block, these commands are executed in one of the following two ways:

(1) The move command and tool function command are started at the same time.

(2) After the move command ends, the tool function command is started.

Explanation

A numeric value following the T code specifies selection of a tool. Part of the numeric value is also used as a tool offset number for specifying an offset value such as a tool offset value. Tools are selected according to the specification method and parameter setting as Table 11.1 (a):

Table 11.1 (a)

Meaning of T code (Note 1) Bit 1 (LGN) of parameter No. 5002 =0	Meaning of T code (Note 1) Bit 1 (LGN) of parameter No. 5002 =1	Parameter setting and offset No. specification method (Note 2)
T XXXXXXX X ↑ ↑ Tool selection / Tool geometry/ tool wear offset	T XXXXXXX X ↑ ↑ Tool selection / Tool wear offset Tool geometry offset	Tool wear offset No. is specified by lowest 1 digit of T code. When parameter No. 5028 is set to 1
T XXXXXX XX ↑ ↑ Tool selection / Tool geometry/ tool wear offset	T XXXXXX XX ↑ ↑ Tool selection / Tool wear offset Tool geometry offset	Tool wear offset No. is specified by lowest 2 digits of T code. When parameter No. 5028 is set to 2
T XXXXX XXX ↑ ↑ Tool selection / Tool geometry/ tool wear offset	T XXXXX XXX ↑ ↑ Tool selection / Tool wear offset Tool geometry offset	Tool wear offset No. is specified by lowest 3 digits of T code When parameter No. 5028 is set to 3

NOTE

1 The maximum number of digits of the T code can be specified in parameter No. 3032. (1 to 8 digits)

2 If 0 is set in parameter No. 5028, the length of the offset number specified in the T code depends on the number of tool offsets.

Example: When the number of tool offsets ranges from 1 to 9: Lowest 1 digit
When the number of tool offsets ranges from 10 to 99: Lowest 2 digits
When the number of tool offsets ranges from 100 to 999: Lowest 3 digits

11.1.1 Tool Offset

T

Overview

Tool offset is used to compensate for the difference when the tool actually used differs from the imagined tool used in programming (usually, standard tool).Tool nose moves on the programmed path by this compensation.

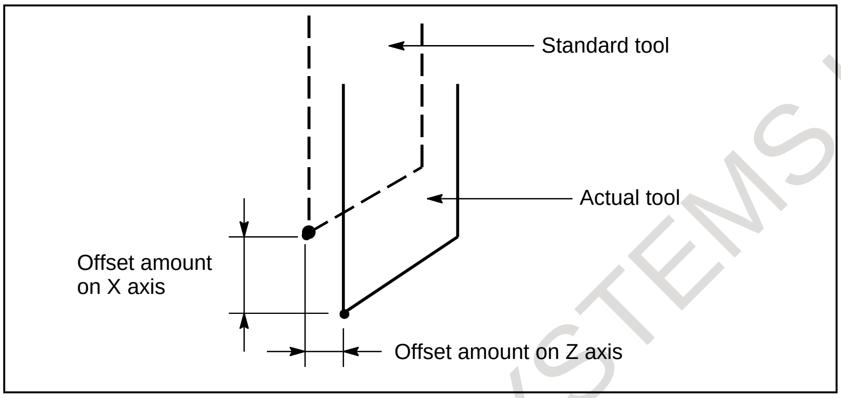


Fig. 11.1.1 (a) Tool offset

11.1.2 Tool Geometry Offset and Tool Wear Offset

T

Overview

Tool geometry offset and tool wear offset are possible to divide the tool offset to the tool geometry offset for compensating the tool shape or tool mounting position and the tool wear offset for compensating the tool nose wear. When these values are not distinguished from each other, the total of the values is set as the tool offset value.
By setting the bit 0 (WOF) of parameter No.3290, the modification of tool wear offset values or tool position offset values by MDI key input can be disabled.
By setting the bit 1 (GOF) of parameter No.3290, the modification of tool geometry offset values by MDI key input can be disabled. In order to permit an operator only to modify tool wear offset values, set the bit 1 (GOF) of parameter No.3290 to 1.

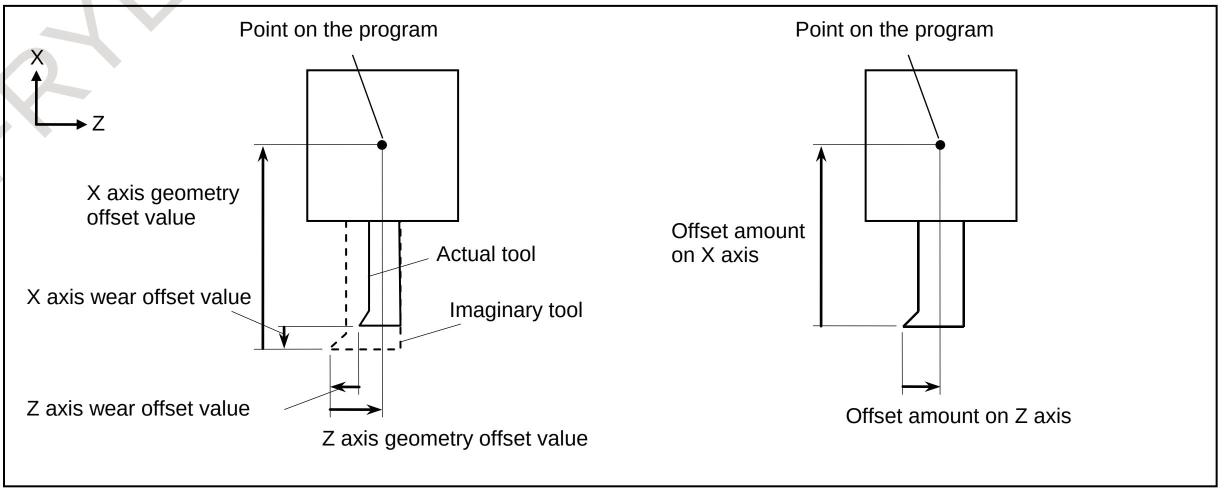


Fig. 11.1.2 (a) If tool geometry compensation and tool wear compensation are distinguished from each other (left) and if they are not (right)

Set the distance from tool nose to the point on the program (tool nose of standard tool, or tool mounting position) to a tool geometry offset value.

Generally, a sign of tool wear offset value is the opposite to a sign of tool geometry offset value. In the example of the above figure, the geometry offset value is a positive and the wear offset value is a negative.

11.1.3 Offset

T

Explanation

- Compensation methods

Two methods are available to geometry compensation and wear compensation, compensation with tool movement and compensation with coordinate shift. Which compensation method to select can be specified with bits 2 (LWT) and 4 (LGT) of parameter No. 5002. When the tool geometry/wear compensation is not enabled (bit 6 (NGW) of parameter No. 8136 is set to 1), however, compensation with tool movement is assumed unconditionally.

Table 11.1.3 (a)

Tool geometry and wear compensation NGW(No.8136#6)	Compensation element	Parameter			
		LWT=0 LGT=0	LWT=1 LGT=0	LWT=0 LGT=1	LWT=1 LGT=1
Disable (NGW=1)	Wear and geometry not distinguished	Tool movement			
Enable (NGW=0)	Wear compensation	Tool movement	Coordinate shift	Tool movement	Coordinate shift
	Geometry compensation	Coordinate shift	Coordinate shift	Tool movement	Tool movement

- Compensation with tool movement

The tool path is offset by the X, Y, and Z tool offset values for the programmed path. The tool offset distance corresponding to the number specified by the T code is added to or subtracted from the end position of each programmed block.

The vector with tool offset X, Y, and Z is called the offset vector. Compensation is the same as the offset vector.

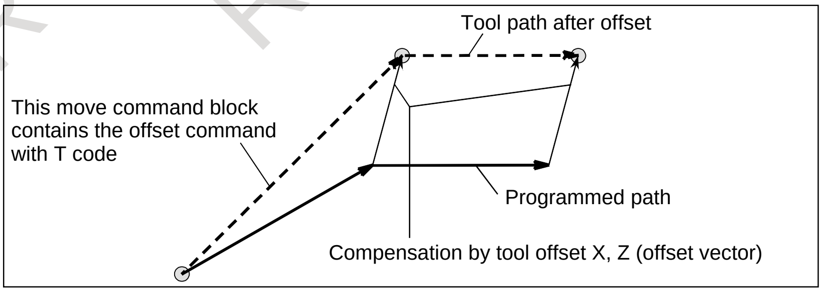


Fig. 11.1.3 (a) Offset operation with tool movement

NOTE

1 When G50 X_Z_T_ ; is specified, the tool is not moved.
The coordinate system in which the coordinate value of the tool position is (X,Z) is set. The tool position is obtained by subtracting the offset value corresponding to the tool offset number specified in the T code.

2 The G codes in the 00 group other than G50 must not be specified in the same block as that containing a T code. If any one of G28, G29, G30, and G53 is specified in the same block together with a T code, alarm PS0245, "T-CODE NOT ALLOWED IN THIS BLOCK", is issued.

- Compensation with coordinate shift

The work coordinate system is shifted by the X, Y, and Z tool offset amounts. Namely, the offset amount corresponding to the number designated with the T code is added to or subtracted from the absolute coordinates.

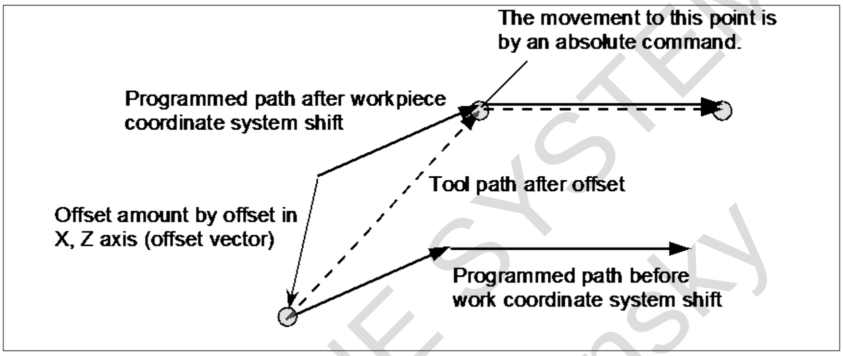


Fig. 11.1.3 (b) Offset operation with coordinate shift

- Starting and canceling offset by specifying a T code

Specifying a tool offset number with a T code means to select the tool offset value corresponding to it and to start offset. Specifying 0 as a tool offset number means to cancel offset.
For offset with tool movement, whether to start or cancel the offset can be specified with bit 6 (LWN) of parameter No. 5002.
For compensation with coordinate shift, the offset is started and canceled when a T code is specified. For the cancellation of geometry compensation, its operation can be selected with bit 5 (LGC) of parameter No. 5002.

Table 11.1.3 (b)

Compensation method	Bit 6 (LWN) of parameter No. 5002=0	Bit 6 (LWN) of parameter No. 5002=1
Tool movement	When a T code is specified	When an axial movement is specified
Coordinate shift	When a T code is specified (Note that geometry compensation can be canceled only if bit 5 (LGC) of parameter No. 5002 = 1.)	

- Canceling offset with reset

Tool offset is canceled under one of the following conditions:
<1> The power to the CNC is turned off and turned back on
<2> The reset button on the MDI unit is pressed
<3> A reset signal is input from the machine to the CNC
In cases <2> and <3> above, it is possible to select a cancel operation using bits 3 (LVC) of parameter No. 5006 and 7 (TGC) of parameter No. 5003.

Table 11.1.3 (c)

Compensation method		Parameter			
		LVC=0 TGC=0	**LVC=1 TGC=0**	**LVC=0 TGC=1**	**LVC=1 TGC=1**
Tool movement	Wear compensation	Not canceled.	Canceled. (When axial movement is specified)	Not canceled.	Canceled. (When axial movement is specified)
	Geometry compensation				
Coordinate shift	Wear compensation	Not canceled.	Canceled.	Not canceled.	Canceled.
	Geometry compensation	Not canceled.	Not canceled.	Canceled.	Canceled.

Notes and restrictions

- Helical interpolation (G02, G03)
  Tool position offset cannot be specified in a block in which helical interpolation is used.
- Coordinate system rotation (G68.1)
  Coordinate system rotation is executed on the command program first, followed by tool position offset.
- 3-dimensional coordinate conversion (G68.1)
  If tool position offset is used, tool position offset with coordinate shift cannot be used. Offset with tool movement must be specified inside a nest of 3-dimensional coordinate conversion. At the beginning block of the 3-dimensional coordinate conversion, tool does not move for a converted tool offset value.
  Example)

```
G68.1 ... ;
T0101;
  :
T0100;
G69.1 ... ;
```

- Workpiece coordinate system preset (G50.3)
  When the bit 7(WTC) of parameter No.1205 is 0, performing workpiece coordinate system preset causes tool position offset with tool movement to be canceled; this does not cause tool position offset with coordinate shift to be canceled.
  When the bit 7(WTC) of parameter No.1205 is 1, performing workpiece coordinate system preset does not cause neither tool position offset with tool movement nor tool position offset with coordinate shift to be canceled.
- Machine coordinate system setting (G53), reference position return (G28), second, third, and fourth reference position return (G30), and manual reference position return

Basically, before performing these commands or operations, cancel tool position offset. These operations do not cause tool position offset to be canceled. The actions in the Table 11.1.3 (d) take place:

Table 11.1.3 (d)

	When the command or operation is specified	When the next axial movement command is specified
Tool movement	The tool offset value is temporarily canceled.	The tool offset value is reflected.
Coordinate shift	Coordinates with the tool offset value reflected are assumed.	Coordinates with the tool offset value reflected are assumed.

- Arbitrary speed threading
  The alarm PS0528, “THREADING FORMAT ERROR” is issued when a T-code and the M-code for starting arbitrary speed threading are specified in the same block.

- Auxiliary function output in moving axis (G50.9)
  The alarm PS5330, “G50.9 FORMAT ERROR” is issued when a T-code and auxiliary function output in moving axis (G50.9) are specified in the same block.

- Programmable parameter input (G10)
  The alarm PS1144, “G10 FORMAT ERROR” is issued when a T-code is specified during programmable parameter input.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1205	WTC							

[Input type] Parameter input
[Data type] Bit path

#7 WTC When workpiece coordinate system preset is done, actual tool position offset by tool movement is :
0: Not considered.
1: Considered.

3032	Allowable number of digits for the T code

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 8
Set the allowable numbers of digits for the T codes.
When 0 is set, the allowable number of digits is assumed to be 8.

	#7	#6	#5	#4	#3	#2	#1	#0
3129							DAP	DRP

[Input type] Parameter input
[Data type] Bit path

#0 DRP For relative coordinate display:
0: The actual position considering a tool offset (tool movement) is displayed.
1: The programmed position excluding a tool offset (tool movement) is displayed.

#1 DAP For absolute coordinate display:
0: The actual position considering a tool offset (tool movement) is displayed.
1: The programmed position excluding a tool offset (tool movement) is displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3290							GOF	WOF

[Input type] Parameter input
[Data type] Bit path

#0 WOF Setting the tool offset value (tool wear offset) by MDI key input is:
0: Not disabled.
1: Disabled. (With parameters Nos. 3294 and 3295, set the offset number range in which updating the setting is to be disabled.)

> **NOTE**
> When tool offset memory A is selected with the M series, the tool offset set in this parameter WOF is followed even if geometric compensation and wear compensation are disabled (bit 6 (NGW) of parameter No. 8136 is set to 1) with the T series.

#1 GOF Setting the tool geometry offset value by MDI key input is:
0: Not disabled.
1: Disabled. (With parameters Nos. 3294 and 3295, set the offset number range in which updating the setting is to be disabled.)

3294	Start number of tool offset values whose input by MDI is disabled

3295	Number of tool offset values (from the start number) whose input by MDI is disabled

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 999

When the modification of tool offset values by MDI key input is to be disabled using bits 0 (WOF) and 1 (GOF) of parameter No. 3290, parameters Nos. 3294 and 3295 are used to set the range where such modification is disabled. In parameter No. 3294, set the offset number of the start of tool offset values whose modification is disabled. In parameter No. 3295, set the number of such values. In the following cases, however, none of the tool offset values may be modified:
- When 0 or a negative value is set in parameter No. 3294
- When 0 or a negative value is set in parameter No. 3295
- When a value greater than the maximum tool offset number is set in parameter No. 3294

In the following case, a modification to the values ranging from the value set in parameter No. 3294 to the maximum tool offset number is disabled:
- When the value of parameter No. 3294 added to the value of parameter No. 3295 exceeds the maximum tool offset number

When the offset value of a prohibited number is input through the MDI panel, the warning "WRITE PROTECT" is issued.

[Example] When the following parameter settings are made, modifications to both of the tool geometry offset values and tool wear offset values corresponding to offset numbers 51 to 60 are disabled:
- Bit 1 (GOF) of parameter No. 3290 = 1 (to disable tool geometry offset value modification)
- Bit 0 (WOF) of parameter No. 3290 = 1 (to disable tool wear offset value modification)
- Parameter No. 3294 = 51
- Parameter No. 3295 = 10

If the setting of bit 0 (WOF) of parameter No. 3290 is set to 0 without modifying the other parameter settings above, tool geometry offset value modification only is disabled, and tool wear offset value modification is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5002	WNP	LWM	LGC	LGT	ETC	LWT	LGN	

[Input type] Parameter input
[Data type] Bit path

#1 LGN Geometry offset number of tool offset :
0: Is the same as wear offset number
1: Specifies the geometry offset number by the tool selection number

> **NOTE**
> This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No.8136 is 0).

#2 LWT Tool wear compensation is performed by:
0: Moving the tool.
1: Shifting the coordinate system.

> **NOTE**
> This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No.8136 is 0).

#3 ETC When a T-code command is two digits or shorter, the T code is:
0: Not extended.
1: Extended.
When this parameter is 1, two-digit or shorter T-code commands are extended. (Three-digit or longer T-code commands are not extended.) The value after extension is determined by the setting of the number of digits in the offset number in T-code commands (parameter No. 5028).

Parameter No. 5028	Number of digits after extension	Sample extension
1	Extended to two digits	Before extension: T1 → After extension: T11
2	Extended to four digits	Before extension: T1 → After extension: T0101
3 or greater	Not extended	

[Example]
- Parameter No. 5028 : 2
- Parameter No. 3032 : 4 (Allowable number of digits in T code)

Before extension	→	After extension
T1	→	T0101 (1-digit command is extended to 4 digits.)
T12	→	T1212 (2-digit command is extended to 4 digits.)
T112	→	T112 (Not extended)
T1122	→	T1122 (Not extended)

> **NOTE**
> 1 The setting of the allowable number of digits in T code (parameter No. 3032) indicates the number of digits in a specified command (before being extended). If the number of digits in the command exceeds the allowable number of digits in T code, the alarm PS0003, “TOO MANY DIGIT” is issued.

> **NOTE**
> 2 This parameter is dedicated to the lathe system. Tool change is available with the turret type setting (bit 3 (TCT) of parameter No. 5040 = 0).
> 3 If the number of digits in the offset number in a T-code command (parameter No. 5028) is set to 0, the value after extension is determined by the number of digits in the number of tool compensation values (parameter No. 5024).
> 4 Common variable #149 for calling a T-code macro is set to the pre-extension value.

#4 LGT Tool geometry compensation is:
0: Compensated by the shift of the coordinate system
1: Compensated by the tool movement

> **NOTE**
> This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No.8136 is 0).

#5 LGC When tool geometry compensation is based on coordinate shifting, the tool geometry offset is:
0: Not canceled by a command with offset number 0.
1: Canceled by a command with offset number 0.

> **NOTE**
> This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No.8136 is 0).

#6 LWM Tool offset operation based on tool movement is performed:
0: In a block where a T code is specified.
1: Together with a command for movement along an axis.

#7 WNP Imaginary tool tip number used for tool nose radius compensation, when the tool geometry/wear compensation function is equipped, is the number specified by:
0: Geometry offset number
1: Wear offset number

	#7	#6	#5	#4	#3	#2	#1	#0
5003	TGC							

[Input type] Parameter input
[Data type] Bit path

#7 TGC A tool geometry offset based on a coordinate shift is:
0: Not canceled by reset.
1: Canceled by reset.

> **NOTE**
> This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No.8136 is 0).

	#7	#6	#5	#4	#3	#2	#1	#0
5004							ORC	

[Input type] Parameter input
[Data type] Bit path

#1 ORC The setting of a tool offset value is corrected as:
0: Diameter value
1: Radius value

NOTE
This parameter is valid only for an axis based on diameter specification. For an axis based on radius specification, specify a radius value, regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5006					LVC		TGC	

[Input type] Parameter input
[Data type] Bit

#1 TGC If a T code is specified in a block where G50, G04, or G10 is specified:
0: No alarm is issued.
1: The alarm PS0245, “T-CODE NOT ALLOWED IN THIS BLOCK” is issued.

#3 LVC A tool offset (geometry/wear) based on a tool movement and wear offset based on a coordinate shift are:
0: Not canceled by reset.
1: Canceled by reset.

5013	Maximum value of tool wear compensation

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (offset unit)
[Min. unit of data] The increment system of a tool offset value is followed.
[Valid data range] The settings of bits 3 to 0 (OFE, OFD, OFC, and OFA) of parameter No. 5042 are followed.

For metric input

OFE	OFD	OFC	OFA	Valid data range
0	0	0	1	0 to 9999.99mm
0	0	0	0	0 to 9999.999mm
0	0	1	0	0 to 9999.9999mm
0	1	0	0	0 to 9999.99999mm
1	0	0	0	0 to 999.999999mm

For inch input

OFE	OFD	OFC	OFA	Valid data range
0	0	0	1	0 to 999.999inch
0	0	0	0	0 to 999.9999inch
0	0	1	0	0 to 999.99999inch
0	1	0	0	0 to 999.999999inch
1	0	0	0	0 to 99.9999999inch

This parameter sets the maximum allowable tool wear compensation value. If an attempt is made to set a tool wear compensation value, the absolute value of which exceeds the value set in this parameter, the following alarm or warning is output:

Input from MDI	Warning: DATA IS OUT OF RANGE
Input by G10	Alarm PS0032: ILLEGAL OFFSET VALUE IN G10.

When 0 or a negative value is set, no maximum allowable value is applied.

[Example] When 30.000 is set
As a tool offset value, a value from -30.000 to +30.000 can be input.

5014 **Maximum value of incremental input for tool wear compensation**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (offset unit)
[Min. unit of data] The increment system of a tool offset value is followed.
[Valid data range] The settings of bits 3 to 0 (OFE, OFD, OFC, and OFA) of parameter No. 5042 are followed.

For metric input

OFE	OFD	OFC	OFA	Valid data range
0	0	0	1	0to9999.99mm
0	0	0	0	0to9999.999mm
0	0	1	0	0to9999.9999mm
0	1	0	0	0to9999.99999mm
1	0	0	0	0to999.999999mm

For inch input

OFE	OFD	OFC	OFA	Valid data range
0	0	0	1	0to999.999inch
0	0	0	0	0to999.9999inch
0	0	1	0	0to999.99999inch
0	1	0	0	0to999.999999inch
1	0	0	0	0to99.9999999inch

Set the maximum allowable value for the tool wear compensation value, input as an incremental value. If the incremental input value (absolute value) exceeds the set value, the following alarm or warning message is output:

Input from MDI	Warning: DATA IS OUT OF RANGE
Input by G10	Alarm PS0032: ILLEGAL OFFSET VALUE IN G10.

When 0 or a negative value is set, no maximum allowable value is applied.

5024 **Number of tool compensation values**

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 999
Set the maximum allowable number of tool compensation values used for each path.
Ensure that the total number of values set in parameter No. 5024 for the individual paths is within the number of compensation values usable in the entire system. The number of compensation values usable in the entire system depends on the option configuration.

If the total number of values set in parameter No. 5024 for the individual paths exceeds the number of compensation values usable in the entire system, or 0 is set in parameter No. 5024 for all paths, the number of compensation values usable for each path is a value obtained by dividing the number of compensation values usable in the entire system by the number of paths.
Tool compensation values as many as the number of compensation values used for each path are displayed on the screen. If tool compensation numbers more than the number of compensation values usable for each path are specified, alarm PS0115, "VARIABLE NO. OUT OF RANGE" is issued.
For example, 64 tool compensation sets are used, 20 sets may be allocated to path 1, 30 sets to path 2, and 14 sets to path 3. All of 64 sets need not be used.

5028	Number of digits of an offset number used with a T code command

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 4

Specify the number of digits of a T code portion that is used for a tool offset number (wear offset number when the tool geometry/wear compensation function is used).
When 0 is set, the number of digits is determined by the number of tool compensation values.
When the number of tool compensation values is 1 to 9: Lower 1 digit
When the number of tool compensation values is 10 to 99: Lower 2 digits
When the number of tool compensation values is 100 to 999: Lower 3 digits

[Example] When an offset number is specified using the lower 2 digits of a T code, set 2 in parameter No. 5028.
Txxxxxx yy
xxxxxx : Tool selection
yy : Tool offset number

In tool management function, set 4 in this parameter to set 4-digit number in parameter No.13265.

> **NOTE**
> A value longer than the setting of parameter No. 3032 (allowable number of digits of a T code) cannot be set.

	#7	#6	#5	#4	#3	#2	#1	#0
5040								OWD

[Input type] Parameter input
[Data type] Bit path

#0 OWD In radius programming (bit 1 (ORC) of parameter No. 5004 is set to 1),
0: Tool offset values of both geometry compensation and wear compensation are specified by radius.
1: Tool offset value of geometry compensation is specified by radius and tool offset value of wear compensation is specified by diameter, for an axis of diameter programming.

> **NOTE**
> This parameter is valid when tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No.8136 is 0).

Alarm and message

Number	Message	Description
PS0245	T-CODE NOT ALLOWED IN THIS BLOCK	One of the G codes, G50, G10, G04, G28, G29, G30, and G53, which cannot be specified in the same block as a T code, was specified with a T code.

11.1.4 Extended Tool Selection Function

T

Overview

In lathe system machines, tools are changed mainly with the following two methods:

(1) With a turret holding multiple tools, tools are changed by turning the turret (T command).
(2) With an automatic tool changer (ATC), tools are changed by using both cartridge indexing (T command) and tool change (such as M06) commands.

To support the tool change method explained in (2) above, the following tool selection specifications apply to this function:
<1> Tool compensation by a T command is disabled. This means that the T command performs auxiliary functions only.
<2> Tool compensation is enabled by using a G code instead of the T command. In this case, the following types of tool compensation are enabled:
- Tool length compensation
- Tool offset (compensation equivalent to that of a T command in the case of turret rotation type)

Explanation

- Selecting a tool change method

Bit 3 (TCT) of parameter No. 5040 selects a tool change method.
This changes the way for specifying tool compensation.
This parameter setting has influence on the range in the Table 11.1.4 (a) :

Table 11.1.4 (a)

	Bit 3 (TCT) of parameter No. 5040 = 0 (Turret type)	Bit 3 (TCT) of parameter No. 5040 = 1 (ATC type)
Operation of T command	Auxiliary function (tool change) and tool compensation	Auxiliary function (tool indexing) only
Tool compensation	Specified with T code	Specified with G43.7 D_
Compensation No. of tool compensation	Specified with T code	Specified with D code
Compensation No. of tool-nose radius compensation	Specified with T code	Specified with D code
Command such as G43, etc. (group 23)	Disabled (alarm)	Enabled
Compensation number of G43, etc. (group 23)	-	Specified with D code

- T command operation

(1) When bit 3 (TCT) of parameter No. 5040 is 0
The T command performs an auxiliary function and tool compensation.
The code signal issued to the machine is the T command value except the last 1 to 3 digits.
For example, suppose that the following command is issued when 2 is set in parameter No. 5028 (the number of digits comprising the offset number in the T code command):
T0313 ;
The T code signal sent to the machine indicates 03.
The tool compensation number, indicated by the lowest 2 digits, is 13.

(2) When bit 3 (TCT) of parameter No. 5040 is 1
The T command performs an auxiliary function only.
In this case, the code signal issued to the machine is the T command value.
For example, suppose that the following is specified:
T0313 ;
The T code signal issued to the machine is 0313. The T code signal is not affected by the setting of parameter No. 5028 (the number of digits comprising the offset number in the T code command).
The tool compensation number is not affected by this command.

- Tool compensation

(1) When bit 3 (TCT) of parameter No. 5040 is 0
The T command performs tool compensation.

(2) When bit 3 (TCT) of parameter No. 5040 is 1
The T command does not perform tool compensation.
To provide tool compensation, specify G43.7 D_.
Tool compensation is performed in the same way as with the T command when bit 3 (TCT) of parameter No. 5040 is 0.

- Compensation number of tool compensation

(1) When bit 3 (TCT) of parameter No. 5040 is 0
The lowest 1 to 3 digits of the T command are used. The number of digits is set in parameter No. 5028.
When tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No.8136 is 0), tool geometry compensation and wear compensation may also be assigned different compensation values by setting bit 1 (LGN) of parameter No. 5002.
For details, see Section "TOOL FUNCTIONS OF LATHE SYSTEM".

(2) When bit 3 (TCT) of parameter No. 5040 is 1
Except that a compensation number is specified with a D command, the same operation as (1) above is performed.

- Compensation number of tool-nose radius compensation

Same as for tool compensation described above.

- Specification of G43, etc.

(1) When bit 3 (TCT) of parameter No. 5040 is 0
G codes of group 23 such as G43 cannot be specified.
Specifying such a G code results in an alarm PS0366, "IMPROPER G-CODE IN TURRET METHOD".

(2) When bit 3 (TCT) of parameter No. 5040 is 1
G codes of group 23 such as G43 can be specified. The following G codes can be specified:
G43/G44: Tool length compensation
G43.7: Tool compensation
Specify a compensation number with D_. The D code is specified in the same way as for tool compensation and tool-nose radius compensation.

The reverse meanings can be set for G43/G44 and G43.7 by setting bit 4 (TLG) of parameter No. 5040.

- Tool compensation memory

As the compensation amount used by the functions listed below, only the value set in Z on the tool compensation memory screen is used, Values such as values set for the X, R, and Y-axis offsets are ignored.

G43/G44: Tool length compensation

As the compensation amount used in the following function, the values set for the X, Z, and Y-axis offsets on the tool compensation memory screen are used:

G43.7: Tool compensation

Limitation

- Switching between tool compensation mode and another compensation mode

In the tool compensation (G43.7) mode, a command such as a tool length compensation command (G43/G44) cannot be specified. Similarly, in a mode such as the tool length compensation mode (G43/G44), tool compensation (G43.7) cannot be specified.
Specifying such a command results in the alarm PS0368, “OFFSET REMAIN AT OFFSET COMMAND”.
To switch between compensation types, specify G49 to cancel the current compensation mode before setting another compensation mode.

- Multiple repetitive canned cycle

When a multiple repetitive canned cycle is executed with bit 3 (TCT) of parameter No. 5040 set to 1, note the following:

> ⚠ **CAUTION**
>
> 1 When a G71 to G76 command is specified in the FS15 tape format, a value such as the depth of cut is specified with a D code. In this case, a D command specified after G71 to G76 is assumed to be the depth of cut. For example, suppose that the command shown below is specified. In this case, the D command <1> is assumed to be the tool compensation number, and the D command <2> is assumed to be the depth of cut.
>
> Example:
>
> <u>D10</u> G71 P_Q_ U_ W_ <u>D7000</u> F_ S_;
> <1> <2>
>
> 2 In the operation of G71 to G73, a G code such as G43 and a D command specified in the finish figure blocks (the portion enclosed by the sequence numbers specified with P_ and Q_) are ignored, and the compensation amount set when the G71 to G73 block is specified becomes valid.

- Changing bit 3 (TCT) of parameter No. 5040

> ⚠ **WARNING**
>
> Before changing the setting of bit 3 (TCT) of parameter No. 5040, cancel the offset. If the setting is changed while the offset is applied, the subsequent offset operation may not be performed correctly or an alarm PS0368 occurs.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5040				TLG	TCT			

[Input type] Parameter input
[Data type] Bit path

#3 TCT The tool change method is based on:

0: Turret rotation. (Tool change operation is performed with a T command only.)
With a T command, an auxiliary function and tool offset operation are performed.

1: Automatic tool changer (ATC).
(Tool change operation is performed with an M command (such as M06)).

With a T command, an auxiliary function only is performed.
This parameter is valid with a lathe system only.

> ⚠ **WARNING**
> Before changing the setting of this parameter, cancel the offset. If the setting is changed while the offset is applied, the subsequent offset operation may not be performed correctly or an alarm PS0368 occurs.

#4 TLG When tool change operation is performed with the automatic tool changer (when bit 3 (TCT) of parameter No. 5040 is set to 1), tool offset operation is specified by:

0: G43.7.
At this time, G43 and G44 function as G codes for tool length compensation.

1: G43.
At this time, G43.7 and G44.7 function as G codes for tool length compensation.

Alarm and message

Number	Message	Description
PS0366	IMPROPER G-CODE IN TURRET METHOD	When the turret change tools method was selected (bit 3 (TCT) of parameter No. 5040 = 0), G43, or G43.7 was commanded.
PS0368	OFFSET REMAIN AT OFFSET COMMAND	- When the ATC change tools method was selected (bit 3 (TCT) of parameter No. 5040 = 1) during G43 mode, G43.7 was commanded. Or, G43 was commanded during G43.7 mode. - After bit 3 (TCT) of parameter No. 5040 was changed in the state in which a tool offset remained, another tool offset was specified.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (For Lathe System) (B-64694EN-1)	Extended tool selection function

11.1.5 Automatic Alteration of Tool Position Compensation (T Function)

T

Overview

After cutting tools are changed manually, entering the tool number of a new tool enables the previous tool position compensation values (tool geometry and wear compensation) to be changed to those for the new tool.

Details

After changing cutting tools manually, place the CNC in the JOG or HNDL (INC), or REF mode, enter the tool number of a new tool as a manual tool compensation tool number signal MTLN <Gn068, Gn069> or MT8N <Gn525-Gn528> from the PMC, and set the compensation command signal MTLC <Gn067.0> to "1". The CNC will read the tool number and change the previous tool position compensation values to those for the specified tool. Upon completion of the compensation value change, the compensation completion signal MTLA <Fn061.5> is output.

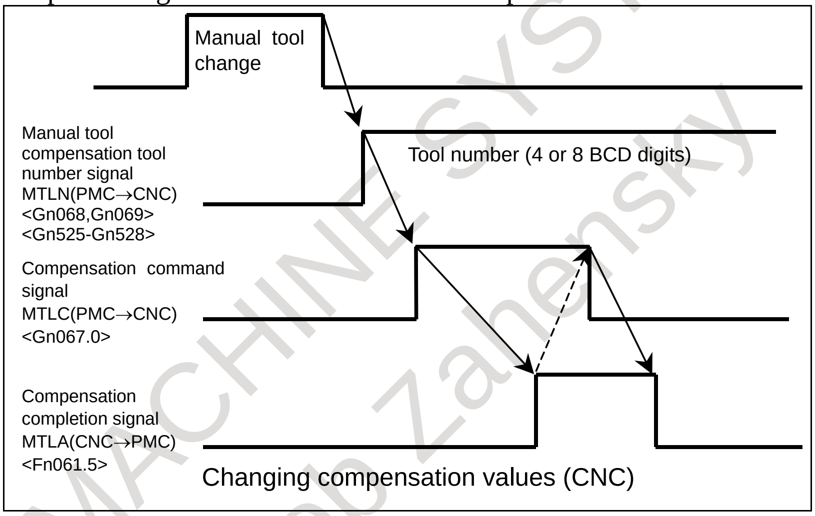


Fig. 11.1.5 (a)

Four BCD digits are used to specify the manual tool compensation tool number MTLN <Gn068, Gn069>. If it is necessary to specify the manual tool compensation tool number with a number exceeding 4 digits, use MT8N <Gn525-Gn528> to specify it with 8 BCD digits. Which size to use can be selected using bit 7 (MTL8D) of parameter No. 11400. The same rules as for the T code command in automatic operation are applied in specifying how many digits, 4 or 8, to use as the geometry or wear compensation numbers (parameter No. 5028).

This function is available only if the CNC is in the JOG, HNDL, or REF mode.

In any other operation mode, during automatic operation (running, pause, or stop), in the tool-nose radius compensation mode, (G41/G42 mode), setting the compensation command signal MTLC to "1" does not enable compensation.

If compensation is unavailable, the compensation completion signal MTLA does not become "1". Instead the compensation uncompleted signal MTLANG <Fn061.4> becomes "1". The reason for MTLANG = 1 can be known from diagnosis data No. 560.

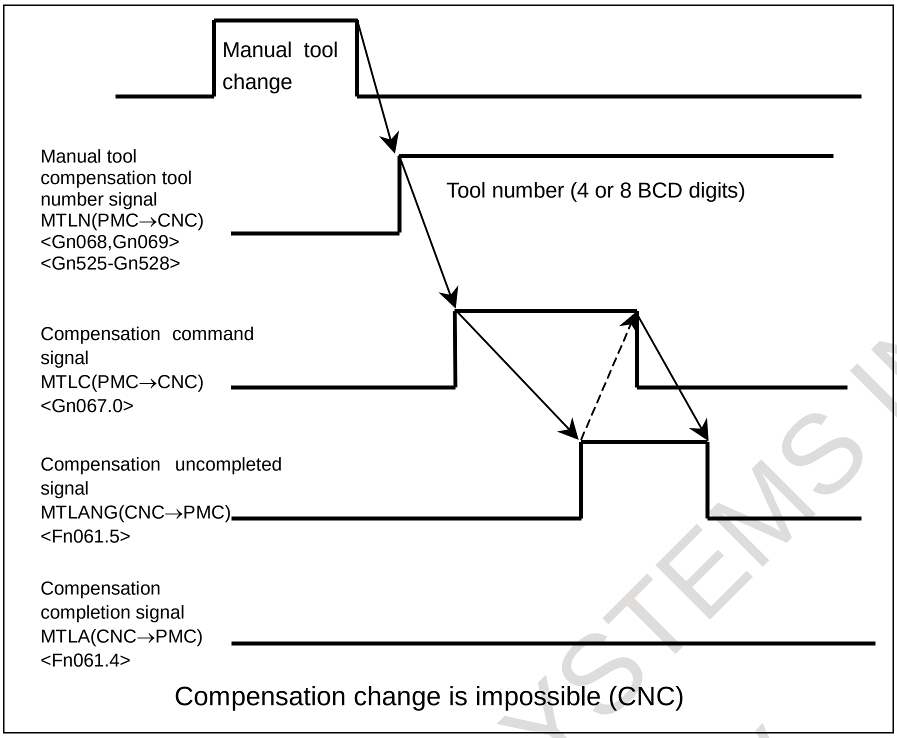


Fig. 11.1.5 (b)

Signal

Manual tool compensation tool number signal (4 digits) MTLN00 to MTLN15<Gn068, Gn069>

[Classification] Input signal

[Function] Specifies a tool number for manual tool compensation, using 4 BCD digits.

Manual tool compensation tool number signal (8 digits) MT8N00 to MT8N31<Gn525-Gn528>

[Classification] Input signal

[Function] Specifies a tool number for manual tool compensation, using 8 BCD digits.

Manual tool compensation command number MTLC<Gn067.0>

[Classification] Input signal

[Function] Requests to perform manual tool compensation. Setting this signal from "0" to "1" causes manual tool compensation to be performed. Re-set it to "0" when the compensation completion signal becomes = "1".

Manual tool compensation completion signal MTLA<Fn061.5>

[Classification] Output signal

[Function] Informs that manual tool compensation is completed. This signal becomes "1" when a compensation value is changed in manual tool compensation. It becomes "0" when the compensation command signal MTLC becomes "0".

Manual tool compensation uncompleted signal MTLANG<Fn061.4>

[Classification] Output signal

[Function] Informs that manual tool compensation is uncompleted. This signal becomes "1" if an attempt to perform manual tool compensation fails in changing a compensation value for any reason. It becomes "0" when the compensation command signal MTLC becomes "0". The reason for the failure of the attempt can be known from diagnosis data No. 560.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn068	MTLN07	MTLN06	MTLN05	MTLN04	MTLN03	MTLN02	MTLN01	MTLN00
Gn069	MTLN15	MTLN14	MTLN13	MTLN12	MTLN11	MTLN10	MTLN09	MTLN08
Gn525	MT8N07	MT8N06	MT8N05	MT8N04	MT8N03	MT8N02	MT8N01	MT8N00
Gn526	MT8N15	MT8N14	MT8N13	MT8N12	MT8N11	MT8N10	MT8N09	MT8N08
Gn527	MT8N23	MT8N22	MT8N21	MT8N20	MT8N19	MT8N18	MT8N17	MT8N16
Gn528	MT8N31	MT8N30	MT8N29	MT8N28	MT8N27	MT8N26	MT8N25	MT8N24
Gn067								MTLC
Fn061			MTLA	MTLANG				

Diagnosis data

0560	Manual tool compensation status number

[Data type] Byte
[Unit of data] None
[Valid data range] 0 to 255

The following numbers are used to indicate whether compensation is completed or the reason for uncompleted compensation (if occur).

0: Manual tool compensation is completed normally.
1: The T code-specified data has exceeded the permissible range.
2: The offset value is out of range.
3: The offset number is out of range.
4: The CNC is undergoing automatic operation or axis movement.
5: The CNC is in the tool-nose radius compensation mode.
6: The CNC is in a mode other than the JOG, HNDL (INC), or REF mode.
7: A CNC parameter has specified an invalid number.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11400								MTL8D

[Input type] Parameter input
[Data type] Bit path

#0 MTL8D The number of digits in the T code used for automatic change of tool position compensation is specified as follows:

0: 4 digits (Existing DI signals <Gn068 and Gn069> are used.)
1: 8 digits (The DI signals <Gn525 to Gn528> are used.)

Notes

If the CNC is in the following conditions, setting the compensation command signal MTLC to “1” cannot change compensation values, and therefore, the compensation uncompleted signal MTLANG becomes “1”.

1) The CNC is in a mode other than the JOG, HNDL (INC), or REF mode.
2) The CNC is at an emergency stop.
3) The reset button on the MDI unit is held pressed.
4) A reset signal is input from the machine side.
5) The CNC is in an alarm condition.
6) The CNC is undergoing automatic operation (start, pause, or stop).
7) The CNC is in the tool-noose radius compensation mode (G41/G42 mode).

With this function, shifted coordinate systems are returned to the previous state by the first T code in the next session of automatic operation, and axis movement occurs by the amount of canceled shifting, thus resuming the position and coordinate systems of the inherent movement type.

Example:

If the amounts of wear compensation are: T1 = 0.1, T2 = 0.2, and T3 = 0.4

1) The state that T1 compensation is applied during automatic operation (compensation of 0.1 axis movement type):
2) The coordinate system is shifted by -0.1 (= T2 - T1) from the state mentioned in step 1 when tool T2 is selected in manual tool compensation.
3) When compensation T3 is applied again during automatic operation, the coordinate system is shifted by 0.2 (= T3 - T2), that is, totally by 0.3. The coordinate system, which has been shifted by -0.1 in step 2, is shifted back to the previous state.

Once the T3 compensation command has been executed, the same position and coordinate system as would be when the T3 compensation command is issued in the T1 compensation state are resumed.

When manual tool compensation ends properly, the T code modal changes to one where manual tool compensation was carried out (the tool function code signals T00 to T31<Fn026 to Fn029> also change).

11.1.6 Y Axis Offset

Overview

When the Y axis, one of the basic three axes, is used with a lathe system, this function performs Y axis offset.
When tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No.8136 is 0), both tool geometry offset and tool wear offset are effective to Y axis offset. This function is enabled when bit 1 (YOF) of parameter No. 8132 is set to 1.

Explanation

Y axis offset results in the same operation as tool offset. For an explanation of the operation, related parameters, and the like, refer to the item “Tool Offset.”

11.1.6.1 Support of arbitrary axes for Y axis offset

Overview

In a lath system, Y axis offset has been usable with the basic three axes only. This function enables Y axis offset to be used with arbitrary axes other than the Y axis, which is one of the basic three axes.

Parameter

5043	**Axis number for which Y-axis offset is used**

[Input type] Parameter input

[Data type] Byte path
[Valid data range] 0 to 24

Set the number of an axis for which Y-axis offset is used.
If 0 or a value beyond the valid data range is set, the Y-axis offset is applied to the Y-axis of the basic three axes. If setting is made for the X- or Z-axis of the basic three axes, the standard tool offset for the X- or Z-axis is not used, and only the Y-axis offset is used.

11.1.7 4th/5th Axis Offset

Overview

This function enables tool offset for the 4th axis and 5th axis following the basic three axes (X, Y, and Z axes). As with tool offsets based on the basic three axes (X, Y, and Z axes), 32 sets of 4th/5th axis tool offsets are available. When tool geometry/wear compensation is enabled (bit 6 (NGW) of parameter No.8136 is 0), tool geometry offset and tool wear offset are enabled. Moreover, the number of sets can be increased from 32 to 64, 99, 400, 999, or 2000.
A tool offset value can be applied to arbitrary axes by setting an axis number (1 to the maximum number of controlled axes) to be used for 4th axis offset in parameter No. 5044 and setting an axis number (1 to the maximum number of controlled axes) to be used for 5th axis offset in parameter No. 5045. By setting bit 1 (NO5) of parameter No. 11400 to 1, 5th axis offset is enabled.
If the same axis as used with the Y axis offset function is specified, however, a Y axis offset value is applied to the specified axis, and a 4th axis offset value or 5th axis offset value is invalidated.
Tool offset values can be input through an I/O device.

Explanation

The operation of 4th/5th axis offset is the same as the operation of tool offset. For the operation, see Section 12.1.1, "TOOL OFFSET".

Setting of tool offset values with the G10 command

By programming, 4th/5th axis offset values can be input.

Format

G10 P_ X_ Y_ Z_ R_ Q_ E_ F_ ;

or

G10 P_ U_ V_ W_ C_ Q_ E_ F_;

P : Offset number
- 0 : Specifies a workpiece coordinate system shift value.
- 1 to 999 : Specifies a tool wear offset value.
- 10000+(1 to 999) : Specifies a tool geometry offset value, with a number (1 to 999) representing an offset number.

X : X axis offset value (absolute)
Y : Y axis offset value (absolute)
Z : Z axis offset value (absolute)
U : X axis offset value (incremental)
V : Y axis offset value (incremental)
W : Z axis offset value (incremental)
R : Tool nose radius compensation value (absolute)
C : Tool nose radius compensation value (incremental)
Q : Virtual tool nose number
E : 4th axis offset value (absolute)
F : 5th axis offset value (absolute)

- Explanation

As in the G10 format above, address E for inputting a 4th axis offset value and address F for inputting a 5th axis offset value allow absolute values only to be input.

> **NOTE**
> 1 When compared with the conventional G10 format for changing tool offset values, address E for specifying a 4th axis offset value and address F for specifying a 5th axis offset value are newly added in the format above.
> 2 When a program based on the G10 format above is executed, only those offset values that correspond to programmed axis addresses and offset numbers are rewritten.

- Address name changing of format

When bit 2 (OFS) of parameter No.11403 is set to 1, the specification address in "Y-axis offset and 4th/5th axis offset" can use axis name 'A' or 'B' instead of default specification addresses 'Y', 'E' and 'F'.

Example) When the following parameters are set, the command address 'A' and 'B' are used for setting of a tool offset.

- Parameter No.5044 is set to 4 (4th axis offset is the 4th axis)
- Parameter No.5045 is set to 5 (5th axis offset is the 5th axis)
- Parameter No.1020 (4th axis ,5th axis) = 65, 66 (Address name of 4th /5th axis offset is 'A' and 'B')
- Bit 2 (OFN) of No.11403 is set to 1 (Address depend on parameter No.1020)

```
%
G10 P01 X_ Z_ A_ B_
G10 P02 X_ Z_ A_ B_
...
G10 P__ X_ Z_ A_ B_
%
```

A_ : Tool compensation data (4th axis offset value in the example).
B_ : Tool compensation data (5th axis offset value in the example).
The other addresses are the same as for the tool compensation amount.

> **NOTE**
> 1 The axis name that can be used as a specification address is only 'A', and 'B'. If 'A' or 'B' is used as the specification address in Y-axis offset, the address 'V' (incremental command of Y-axis offset) cannot be used.
> 2 If the either of following is set, the default specification addresses 'Y', 'E', and 'F' are used.
> - Parameter No.1020 is set to axis name other than 'A(65)' or 'B(66)'.
> - An extended axis name is used.
> 3 When bit 2 (OFN) of parameter No.11403 is set to 1, conventional offset data cannot be read. If conventional offset data is read, set 0 to bit 2 (OFN) of parameter No.11403.

Limitation

1. This function supports the common memory between each path.
2. This function does not support direct input/counter input of tool offset values measured.
3. This function does not support offset value writing using a custom macro variable.

4. This function does not support the PMC window function.
   However, this function supports writing/reading of 4th/5th axis offset values to and from the C Language Executor.
5. This function does not support external data input. So, no 4th/5th axis offset value can be modified with a PMC ladder.
6. This function does not support the second geometry tool offset function.
7. This function does not support the tool management function.
8. This function does not support the wrong operation prevention function.
9. This function does not support the program coordinate switch screen function and the offset memory switch function.

Parameter

5044	Axis number for which 4th-axis offset is used

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0, 1 to number of controlled axes

Set the number of an axis for which the 4th-axis offset is used.
When a value ranging from 1 to the number of controlled axes is set in this parameter, the 4th-axis offset is applied to the set axis number. If 0 or a value beyond the valid data range is set, the 4th-axis offset is not used. For the basic two axes X and Z, the standard tool offsets are used, so the 4th-axis offset cannot be used. When the axis set for the Y-axis offset function is set in this parameter, the Y-axis offset is used for the axis, and the 4th-axis offset is not used.

5045	Axis number for which 5th-axis offset is used

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0, 1 to number of controlled axes

Set the number of an axis for which the 5th-axis offset is used.
When a value ranging from 1 to the number of controlled axes is set in this parameter, the 5th-axis offset is applied to the set axis number. If 0 or a value beyond the valid data range is set, the 5th-axis offset is not used. For the basic two axes X and Z, the standard tool offsets are used, so the 5th-axis offset cannot be used. When the axis set for the Y-axis offset function is set in this parameter, the Y-axis offset is used for the axis, and the 4th-axis offset is not used.
When settings are made so that both the 5th-axis offset and 4th-axis offset apply to the same axis, only the 4th-axis offset is used, and the 5th-axis offset is not used.

	#7	#6	#5	#4	#3	#2	#1	#0
5040	NO4							

[Input type] Parameter input
[Data type] Bit path

#7 NO4 4th axis offset function is:
0: Used.
1: Not used.

	#7	#6	#5	#4	#3	#2	#1	#0
11400							NO5	

[Input type] Parameter input
[Data type] Bit path

#1 NO5 The fifth axis offset function is:
0: Not used.
1: Used.

11.2 TOOL FUNCTIONS OF MACHINING CENTER SYSTEM

M

Overview

Selection of tools can be done by commanding tool numbers with up to an 8-digit numeral after address T.

Signal

Refer to "AUXILIARY FUNCTION/2ND AUXILIARY FUNCTION"

Parameter

3032	Allowable number of digits for the T code

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 8
Set the allowable numbers of digits for the T code.
When 0 is set, the allowable number of digits is assumed to be 8.

Note

NOTE
When a move command and a tool function are specified in the same block, the commands are executed in one of the following two ways:
(i) Simultaneous execution of the move command and tool function commands.
(ii) Executing tool function commands upon completion of move command execution.
The selection of either (i) or (ii) depends on the sequence program of PMC.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Tool selection command Tool selection function
CONNECTION MANUAL (FUNCTION) (This manual)	Auxiliary function

11.2.1 Tool Compensation Memory

M

Overview

Tool compensation values include geometry compensation values and wear compensation values.
Tool compensation values can also be used without discriminating between geometry and wear compensation values.

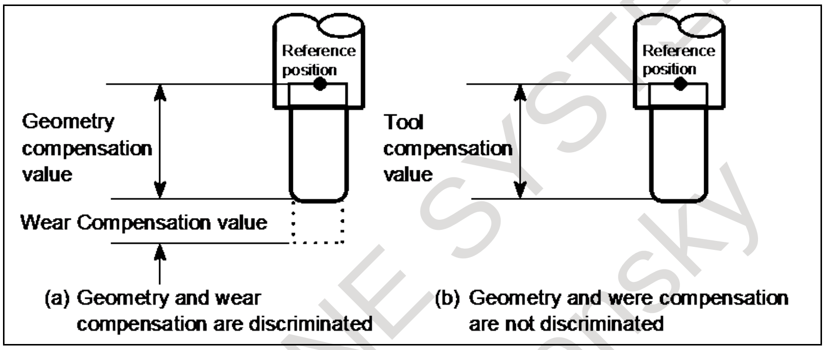


Fig. 11.2.1 (a) Geometric compensation and wear compensation

Tool compensation values can be entered into CNC memory from the MDI or from a program.
A tool compensation value is selected from the CNC memory when the corresponding code is specified after address H or D in a program.
The value is used for tool length compensation, cutter compensation, or the tool offset.

- Unit of range of tool compensation value

A tool offset unit and valid data range can be chosen from the following by setting the parameters:

Table 11.2.1 (a)
Unit of range of tool compensation value (metric input)

OFE	OFD	OFC	OFA	Unit	Range
0	0	0	1	0.01mm	±9999.99mm
0	0	0	0	0.001mm	±9999.999mm
0	0	1	0	0.0001mm	±9999.9999mm
0	1	0	0	0.00001mm	±9999.99999mm
1	0	0	0	0.000001mm	±999.999999mm

Unit of range of tool compensation value (inch input)

OFE	OFD	OFC	OFA	Unit	Range
0	0	0	1	0.001inch	±999.999inch
0	0	0	0	0.0001inch	±999.9999inch
0	0	1	0	0.00001inch	±999.99999inch
0	1	0	0	0.000001inch	±999.999999inch
1	0	0	0	0.0000001inch	±99.9999999inch

- Tool compensation memory

One of the tool compensation memory A/C can be selected according to the configuration of offset amount.

(1) Tool compensation memory A
There is no difference between geometry compensation memory and wear compensation memory in tool compensation memory A. Therefore, amount of geometry offset and wear offset together is set as the offset memory. There is also no differences between cutter compensation (D code) and tool length compensation (H code).

(2) Tool compensation memory C
Memory for geometry compensation and wear compensation is separate in tool compensation memory C. Geometry compensation and wear compensation can thus be set separately. Separate memories are prepared for cutter compensation (for D code) and for tool length compensation (for H code).

The above description is summarized as follows:

Table 11.2.1 (b)

Tool compensation memory	Compensation amount
A	Tool compensation amount (Geometry compensation value + Wear compensation value)
C	Geometry compensation value for H code
	Geometry compensation value for D code
	Wear compensation value for H code
	Wear compensation value for D code

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3109							DWT	

[Input type] Parameter input
[Data type] Bit path

#1 DWT Characters G and W in the display of tool wear/geometry compensation amount
0: The characters are displayed at the left of each number.
1: The characters are not displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3205				OSC				

[Input type] Parameter input
[Data type] Bit

#4 OSC On the offset screen, offset value erasure by a soft key is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3290							GOF	WOF

[Input type] Parameter input
[Data type] Bit path

#0 WOF Setting the tool offset value (tool wear offset) by MDI key input is:
0: Not disabled.

1: Disabled. (With parameters Nos. 3294 and 3295, set the offset number range in which updating the setting is to be disabled.)

> **NOTE**
> When tool offset memory A is selected with the M series, the tool offset value set in this parameter is followed even if geometric compensation and wear compensation are not specified with the T series.

#1 GOF Setting the tool geometry offset value by MDI key input is:

0: Not disabled.

1: Disabled. (With parameters Nos. 3294 and 3295, set the offset number range in which updating the setting is to be disabled.)

3294	Start number of tool offset values whose input by MDI is disabled

3295	Number of tool offset values (from the start number) whose input by MDI is disabled

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 999

When the modification of tool offset values by MDI key input is to be disabled using bits 0 (WOF) and 1 (GOF) of parameter No. 3290, parameters Nos. 3294 and 3295 are used to set the range where such modification is disabled. In parameter No. 3294, set the offset number of the start of tool offset values whose modification is disabled. In parameter No. 3295, set the number of such values. In the following cases, however, none of the tool offset values may be modified:

- When 0 or a negative value is set in parameter No. 3294
- When 0 or a negative value is set in parameter No. 3295
- When a value greater than the maximum tool offset number is set in parameter No. 3294

In the following case, a modification to the values ranging from the value set in parameter No. 3294 to the maximum tool offset number is disabled:

- When the value of parameter No. 3294 added to the value of parameter No. 3295 exceeds the maximum tool offset number

When the offset value of a prohibited number is input through the MDI panel, the warning "WRITE PROTECT" is issued.

[Example] When the following parameter settings are made, modifications to both of the tool geometry offset values and tool wear offset values corresponding to offset numbers 51 to 60 are disabled:

- Bit 1 (GOF) of parameter No. 3290 = 1 (to disable tool geometry offset value modification)
- Bit 0 (WOF) of parameter No. 3290 = 1 (to disable tool wear offset value modification)
- Parameter No. 3294 = 51
- Parameter No. 3295 = 10

If the setting of bit 0 (WOF) of parameter No. 3290 is set to 0 without modifying the other parameter settings above, tool geometry offset value modification only is disabled, and tool wear offset value modification is enabled.

5013	Maximum value of tool wear compensation

[Input type] Parameter input

[Data type] Real path
[Unit of data] mm, inch (offset unit)
[Min. unit of data] The increment system of a tool offset value is followed.
[Valid data range] The settings of bits 3 to 0 (OFE, OFD, OFC, and OFA) of parameter No. 5042 are followed.

For metric input

OFE	OFD	OFC	OFA	Valid data range
0	0	0	1	0 to 9999.99mm
0	0	0	0	0 to 9999.999mm
0	0	1	0	0 to 9999.9999mm
0	1	0	0	0 to 9999.99999mm
1	0	0	0	0 to 999.999999mm

For inch input

OFE	OFD	OFC	OFA	Valid data range
0	0	0	1	0 to 999.999inch
0	0	0	0	0 to 999.9999inch
0	0	1	0	0 to 999.99999inch
0	1	0	0	0 to 999.999999inch
1	0	0	0	0 to 99.9999999inch

This parameter sets the maximum allowable tool wear compensation value. If an attempt is made to set a tool wear compensation value, the absolute value of which exceeds the value set in this parameter, the following alarm or warning is output:

Input from MDI	Warning: DATA IS OUT OF RANGE
Input by G10	Alarm PS0032: ILLEGAL OFFSET VALUE IN G10.

When 0 or a negative value is set, no maximum allowable value is applied.

[Example] When 30.000 is set
As a tool offset value, a value from -30.000 to +30.000 can be input.

5014	Maximum value of incremental input for tool wear compensation

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (offset unit)
[Min. unit of data] The increment system of a tool offset value is followed.
[Valid data range] The settings of bits 3 to 0 (OFE, OFD, OFC, and OFA) of parameter No. 5042 are followed.

For metric input

OFE	OFD	OFC	OFA	Valid data range
0	0	0	1	0to9999.99mm
0	0	0	0	0to9999.999mm
0	0	1	0	0to9999.9999mm
0	1	0	0	0to9999.99999mm
1	0	0	0	0to999.999999mm

For inch input

OFE	OFD	OFC	OFA	Valid data range
0	0	0	1	0to999.999inch
0	0	0	0	0to999.9999inch
0	0	1	0	0to999.99999inch
0	1	0	0	0to999.999999inch
1	0	0	0	0to99.9999999inch

Set the maximum allowable value for the tool wear compensation value, input as an incremental value. If the incremental input value (absolute value) exceeds the set value, the following alarm or warning message is output:

Input from MDI	Warning: DATA IS OUT OF RANGE
Input by G10	Alarm PS0032: ILLEGAL OFFSET VALUE IN G10.

When 0 or a negative value is set, no maximum allowable value is applied.

Alarm and message

Number	Message	Description
PS0032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.

Warning message	Description
DATA IS OUT OF RANGE	The value searched exceeds the permitted range.

11.3 TOOL MANAGEMENT FUNCTION

11.3.1 Tool Management Function

Overview

The tool management function totally manages tool information including information about tool offset and tool life.

Explanation

A tool type number is specified with a T code. The tool type number is any number the user can define freely. With tool type numbers, tools can be grouped by various conditions such as life, compensation value, and cutting conditions. When each type is assumed to have a single tool, tool type numbers are equivalent to unique tool numbers.

For each tool, an information storage area is prepared in the CNC (SRAM). This area contains information such as a tool type number, tool life, tool status (such as a breakage condition), tool compensation number (H, D, G, or W), spindle speed (S), cutting feedrate (F), and freely-definable customize data. Such data is called tool management data.

A cartridge management table that links cartridge information and tool management data is provided so that the CNC can manage the cartridges of the machine and tool change operations. In addition, areas for managing the tools in the spindle and tool standby positions are prepared.

When a tool type number is specified using a T code command, a tool that has the tool type number and the shortest life is searched for, and the cartridge number and pot number of the location where the tool is stored are output to the PMC. Then, a tool change operation using the cartridge number and pot number is enabled by the PMC ladder sequence.

Tool information in the CNC is managed by the tool management data and cartridge management table (including the spindle management table and standby position table).

- Tool management data

As tool management data, information about a tool is registered for each data number.

Tool management function 64 sets	64 sets in total
Tool management function 240 sets	240 sets in total
Tool management function 1000 sets	1000 sets in total

Details of data

The following details the data registered for each data number:

- Tool type number (T code)

Item	Description
Data length	4byte
Valid data range	0,1 to 99,999,999

- Tool life counter

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds)

The value of an increment counter, that is, the number of use times (time) is indicated. The remaining life value is [the maximum tool life value minus tool life counter value].

- Maximum tool life value

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds)

- Notice life value

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds)

If a value other than 0 is set, the tool life expiration notice signal TLCHB <Fn064.3> or TLCHBx <Fn329.4 to 7> is output when the remaining life value of the tool (= maximum tool life value minus tool life counter value) has reached the set value.

- Tool life status

Item	Description
Data length	1byte
Detail data	0: Life management is not performed 1: Tool not yet used 2: Life remains 3: Life expired 4: Tool breakage (skip)

The machine (PMC) determines tool breakage and stores corresponding information through the window. In tool management of the CNC, a broken tool is regarded as being equivalent to tools whose lives have expired.

- Tool information

Item	Description
Data length	1 byte (flag data)
#0 RGS	0: Tool management data is invalid (-) 1: Tool management data is valid (R)

Item	Description
#1 TIM	0: Type for counting the number of use times (C) 1: Type for counting time (T)
#2 BDT	0: Normal tool (N) 1: Oversize tool (B)
#3 LOC	0: Data accessible (U) 1: Data not accessible (L)
#4 SEN	When the tool life status indicates that tool management is not performed: 0: This tool is not searched (-) 1: This tool is searched (S)
#5 to #7	Reserved

When RGS is set to 0 in tool management data, the tool management data is regarded as not being registered even when values are set for other items.

NOTE
Make sure you specify the same life count type for all tools of the same type. You can check tools of the same type for any difference in life count type, using the "check function."

M

- Tool length compensation number (H)

Item	Description
Data length	2byte
Valid data range	0 to 999

- Cutter compensation number (D)

Item	Description
Data length	2byte
Valid data range	0 to 999

T

- Tool geometry compensation number (G)

Item	Description
Data length	2byte
Valid data range	0 to 999

- Tool wear compensation number (W)

Item	Description
Data length	2byte
Valid data range	0 to 999

NOTE
When the machine control type is the combined system type, tool length compensation and cutter compensation numbers are used for paths for the machining center system, and for paths for the lathe system, tool geometry compensation and tool wear compensation numbers are used.

- Spindle speed (S)

Item	Description
Data length	4byte
Unit of data	min^{-1}

Valid data range	1 to 99,999

- Feedrate (F)

Item	Description
Data length	4byte
Unit of data	mm/min, inch/min, deg/min, mm/rev, inch/rev
Valid data range	0 to 99,999,999

As additional tool management data, areas for setting customize data (5 data items including customize data 0 and customize data 1 to 4) are provided without defining specific usage. The user can use these customize data areas freely to set a warning life value, cutting resistance, override value, spindle current value, maximum and minimum S/F, and other items according to the targeted application.

- Customize data 0

Item	Description
Data length	1 byte (bit type)
Valid data range	0 or 1 on a bit-by-bit basis

- Customize data 1 to 4 (to 20) (to 40)

Item	Description
Data length	4byte
Valid data range	-99,999,999 to 99,999,999

- Cartridge management table

The storage status of tools in cartridges is managed with the cartridge management table.

- Multiple cartridge numbers can be defined. (Numbers from 1 to 8, up to eight cartridges)
- The maximum number of pots for all cartridges is 64, 240, or 1000, which depends on whether a tool management data option is selected.
- Cartridge numbers and pot numbers can be assigned freely within cartridge management data by parameter setting (described later).
- The tool management data number adjacent to a pot number is linked with tool data defined in the tool management table. Therefore, the tool attached to the pot is indicated.
- Zero set as a data number indicates that no tool is attached.
- The cartridge management table can be read from and written to through the PMC window and FOCAS2.

The spindle management table and standby position table are provided to indicate special cartridge positions.

- Spindle positions and standby positions, regarded as special cartridge positions, have fixed cartridge numbers 11 to 14 (the positions of the first to fourth spindles) and 21 to 24 (the first to fourth standby positions).
- With the PMC window, the spindle position table and standby position table can be read from and written to.
- Tool life counting is performed only for the tools at the spindle positions.

- Multi-path system

The tool management data and cartridge management table are common data among the paths. The spindle management table and standby position table, however, are treated as independent data for each path.

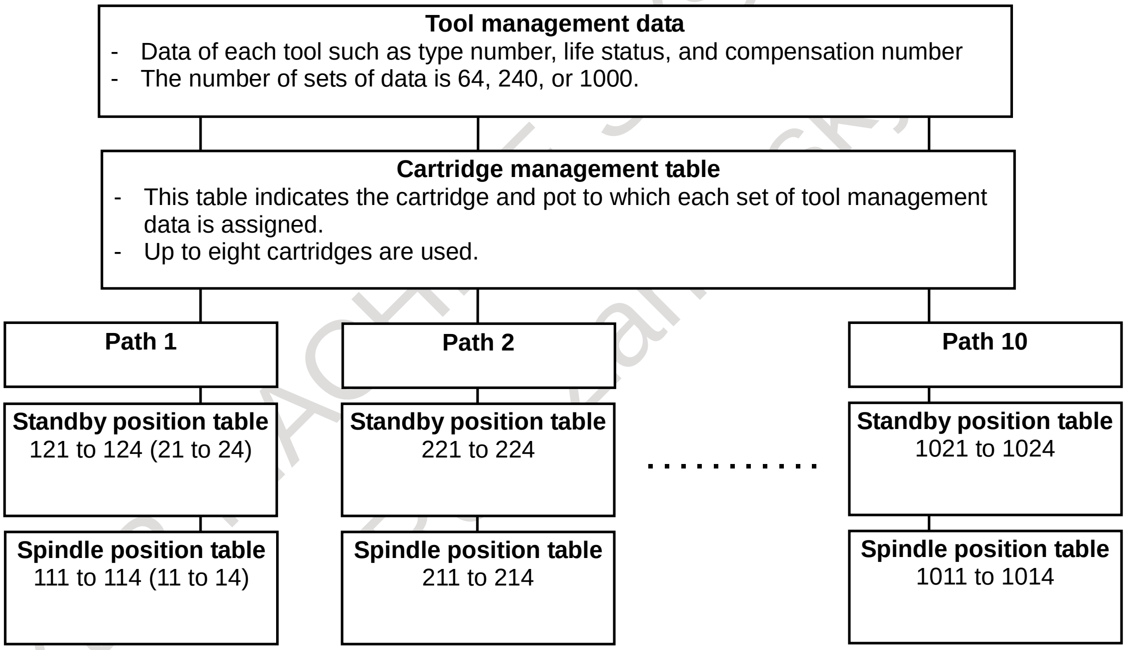
When the spindle table or standby position table is specified as a cartridge from the PMC window, specify the Table 11.3.1 (a), in which the path number is set in the hundred's place:

Table 11.3.1 (a)

	Spindle position			
	First	**Second**	**Third**	**Fourth**
First path	111(11)	112(12)	113(13)	114(14)
Second path	211	212	213	214
Third path	311	312	313	314
Fourth path	411	412	413	414

	Standby position			
	First	**Second**	**Third**	**Fourth**
First path	121(21)	122(22)	123(23)	124(24)
Second path	221	222	223	224
Third path	321	322	323	324
Fourth path	421	422	423	424

NOTE
When specifying 111, 121, and so on to specify the first path, you may specify just 11, 21, and so on.



NOTE
The tool management data and cartridge management table are data common to the M and T series.

- Tool life management and tool change

The CNC performs tool life management, regarding tools having the same tool type number as a group. When a tool type number (T code) is specified by an NC program, the tool management data registered in the CNC is searched to find a tool having the shortest life among the tools with the same tool type number.
The cartridge number and pot number corresponding to the searched tool are output as a T code signal to the PMC. Based on the output cartridge number and pot number, the PMC performs preparation for a tool change (to the next tool). A different tool can also be selected on the PMC side.
The CNC performs tool life counting for each tool that is at a spindle position in the spindle management table.

When the lives of all tools having the tool type number specified by the T code have expired, alarm PS5317, “ALL TOOL LIFE IS OVER” is issued. If there is a tool in the spindle position or standby position, that tool is selected to continue machining.
As the T code signal, a specified tool type number instead of the cartridge number and pot number can also be output directly by parameter setting.
There are two types of tool life management counting methods: counting the number of use times and counting cutting time. One of the counting methods is set in tool information of tool management data.

Other major specifications related to tool life management are as follows:
Tool type number (T code) : Up to 8 digits (1 to 99,999,999)
Maximum tool life value : 99,999,999 times when the number of use times is specified
: 999 hours 59 minutes 59 seconds when time is specified
Life count interval when time is specified : 1 second
Tool life management count restart M code : Enabled
Tool life count override : Enabled

- Examples of performing tool life management

Operation examples of tool life management are given below, in which the tool management data and cartridge management table are set as Table 11.3.1 (b):

Table 11.3.1 (b)
Tool management data

Data No.	Tool type No. (T)	Tool life counter	Maximum tool life value	Tool life status	Tool information	Tool length compensation No. (H)	Cutter compensation No. (D)
1	11111111	985	1000	Not expired	UNCR	1	1
2	11111111	0	1000	Not expired	UNCR	2	2
3	11111111	0	1000	Not expired	UNCR	3	3
4	22222222	0	2000	Not expired	UBCR	31	41

Cartridge management table

No.	Cartridge No.	Pot No.	Tool management data No.
1	1	1	3
2	1	2	12
:	:	:	:
29	1	29	2
30	1	30	1
31	2	1	11
:		:	:
63	2	29	21
64	2	30	0

Spindle management table

Spindle name	Cartridge No. (Spindle No.)	Data No.
Spindle position	11	0

Standby position table

Standby position name	Cartridge No. (Standby position No.)	Data No.
Standby position	21	0

Tool life management is explained below by using the following sample program:

```
Program example
  :
N10 T11111111 ;
  :
```

```
N80 M06 ;
  :
N200 G01 X100.0 F100.0 ;
  :
N999 M30 ;
```

- Selecting a tool having a tool type number and shortest life

The following example explains how a tool having a tool type number specified by a T code is selected:

N10 T11111111 ;

(1) Among tools whose tool type number is 11111111, a tool having the shortest life is searched for(*1). Since the tool with the shortest life is assigned tool management data number 1, the numbers of the cartridge and pot holding this tool are obtained.
    The following tools are not searched:
    - Tools not assigned to any cartridge
    - Tools whose tool life status is 0 (Life management is not performed.)(*2)
    - Tools whose tool life status is 3 (Life has expired.)
    - Tools whose tool life status is 4 (tool breakage)
    - Tools whose tool information bit 0 invalidates the tool management data
    - Tool being edited on the tool management screen
    - Tools whose tool life counter indicates the maximum tool life value.

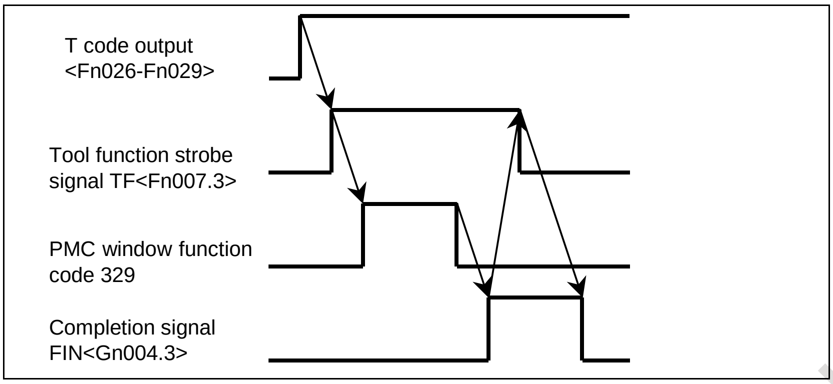
    If multiple tools have the same life value, a search is made according to the following priority:

High	Spindle position
	Standby position
Low	Cartridge

    If a cartridge contains tools having the same life value, the tool with the smaller tool management number takes priority.

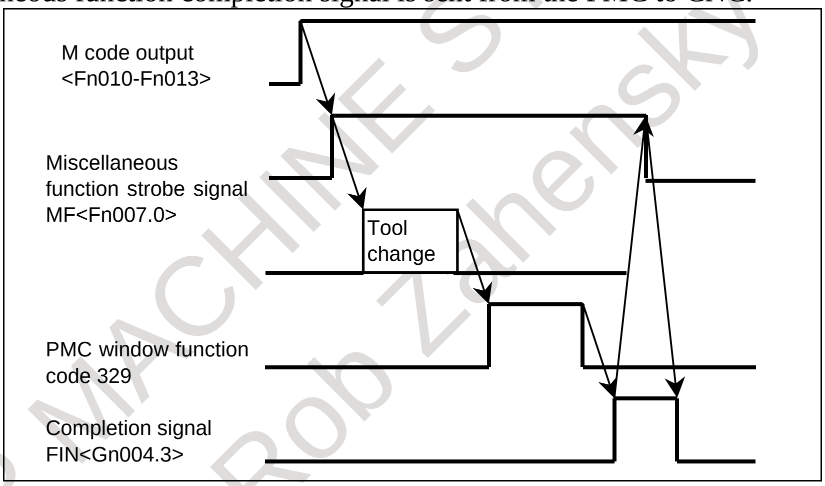
> **NOTE**
> 1 If bit 6 (SWC) of parameter No. 13203 is 1, a tool whose customize data with a customize data number specified in parameter No. 13260 is smallest is searched for instead of a tool with the shortest life.
> 2 Even when the tool life status of a tool shows that life management is not performed (= 0), the tool is targeted for a search operation if bit 4 (SEN) of its tool information is set to 1. In this case, the remaining life is not checked. The cartridge number and pot number of the tool found first are output as the search result.

(2) Since the tool with tool management data number 1 is stored at cartridge number 1 and pot number 30, binary value 0001 is output at PMC addresses <Fn026 to Fn027>, and binary value 0030 is output at <Fn028 to Fn029>.
(3) On the machine side, the tool searched for is moved to the standby position.
(4) By using the PMC window (function code 329), the tool whose cartridge number 1 and pot number 30 in the cartridge management table is moved to the standby position (cartridge number 21).



N80 M06;

(1) In response to the M06 command, M code binary value 0006 is output to PMC F addresses <Fn010 to Fn013> and the MF signal is output to <Fn007.0>.
(2) The machine performs a tool change operation, and moves the tool from the standby position to spindle position.
(3) With the PMC window (function code 329), the tools at the standby position and spindle position in the cartridge management table are changed.
(4) The CNC regards the tool moved to the spindle position (cartridge number 11) by the PMC window (function code 329) as a new tool targeted for tool life management.
(5) The miscellaneous function completion signal is sent from the PMC to CNC.



- When a tool with shortest life is attached at spindle position

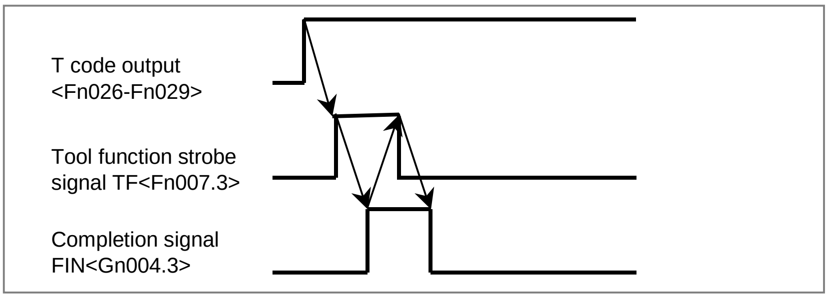
An operation example in which a T code command is issued when a tool with the shortest life is already attached at the spindle position is explained below.

Spindle name	Cartridge No. (Spindle No.)	Data No.
Spindle position	11	1

Standby position name	Cartridge No. (Standby position No.)	Data No.
Standby position	21	0

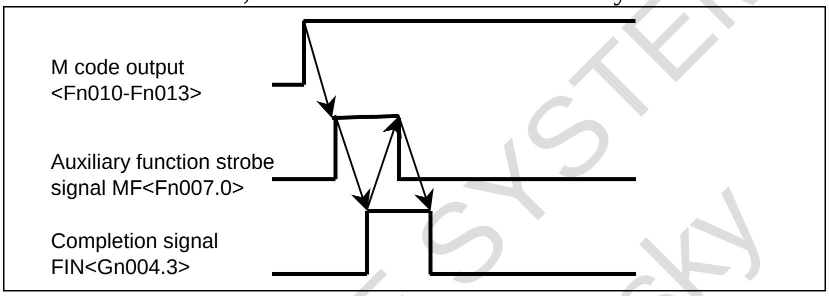
N10 T11111111 ;

(1) From the tools with tool type number 11111111, a tool with the shortest life is searched for.
(2) Since the tool with the shortest life is the tool at the spindle position (tool management data number 1), binary value 0011 is output to PMC addresses <Fn026 to Fn027>. At the addresses indicating the pot number <Fn028 to Fn029>, binary value 0000 is output.
(3) Tool function strobe signal TF is output to the PMC.
(4) Since a tool change operation is not needed, the PMC sends completion signal FIN to the CNC.



N80 M06;

(1) M code binary value 6 is output to the PMC addresses <Fn010 to Fn013>.
(2) Miscellaneous function strobe signal MF <Fn007.0> is output to the PMC.
(3) Since a tool change operation is not needed, the PMC sends completion signal FIN to the CNC.
(4) The CNC starts counting the tool life of the tool held at the spindle position. For the type for counting the number of use times, the life count is incremented by one.



- When the life of the tool being used has expired

An operation example when the tool life of the tool being used (the tool at the spindle position) has expired is explained below.

Tool management data

Data No.	Tool type No. (T)	Tool life counter	Maximum tool life value	Tool life status	Tool information	Tool length compensation No. (H)	Cutter compensation No. (D)
1	11111111	999	1000	Not expired	UNCR	1	1
2	11111111	0	1000	Not expired	UNCR	2	2
3	11111111	0	1000	Not expired	UNCR	3	3

Cartridge management table

No.	Cartridge No.	Pot No.	Tool management data No.
1	1	1	3
2	1	2	12
:	:	:	:
29	1	29	2
30	1	30	0

Spindle management table

Spindle name	Cartridge No. (Spindle No.)	Data No.
Spindle position	11	1

Standby position table

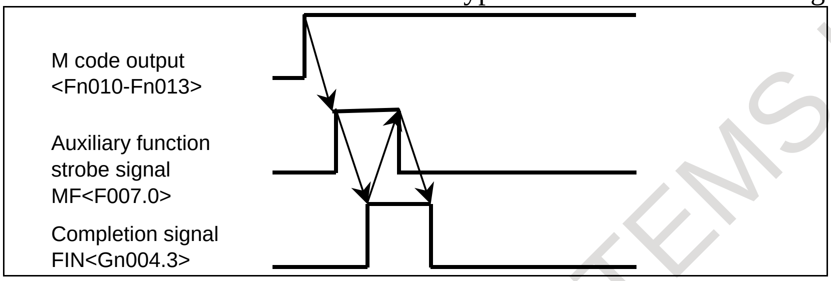
Standby position name	Cartridge No. (Standby position No.)	Data No.
Standby position	21	0

- Type for counting the number of use times

When a life is counted by counting the number of use times, the life count is incremented by one each time M06 (or an M code for restarting tool life counting or T code) is specified.

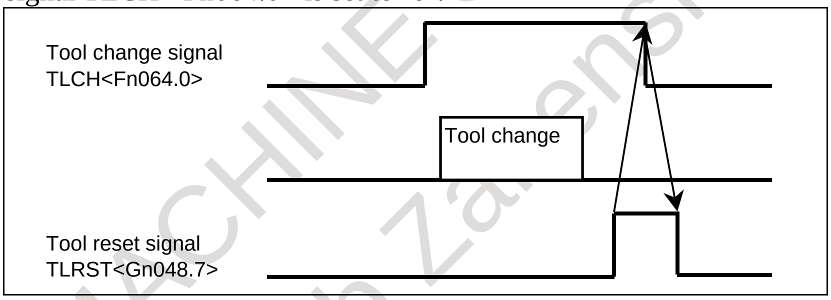
N80 M06;

(1) M code binary value 6 is output to PMC addresses <Fn010 to Fn013>.
(2) Miscellaneous function strobe signal MF <Fn007.0> is output to the PMC.
(3) Since a tool change operation has already completed, no tool change operation is performed.
(4) The PMC sends miscellaneous function completion signal FIN to the CNC.
(5) The CNC increments the life count for the tool held at the spindle position (tool management data No. 1) by one.
(6) The life count reaches 1000 (the maximum tool life), so the tool life expires.
(7) The CNC changes the tool life status for tool management data number 1 to 3 (life has expired).
(8) The CNC searches for a tool that has the same tool type number and has remaining life.



N999 M30;

(1) When the lives of all tools having the same tool type number have expired, tool change signal TLCH <Fn064.0> is set to "1" immediately even during operation.
(2) After a change to a new tool, tool reset signal TLRST <Gn048.7> is set to "1".
(3) Tool change signal TLCH <Fn064.0> is set to "0".



- Type for counting time

Operations performed when a life is counted by counting time are explained below.
Suppose that the tool management data is set as listed below.
Also suppose that the cartridge management table is the same as for the type for counting the number of use times.

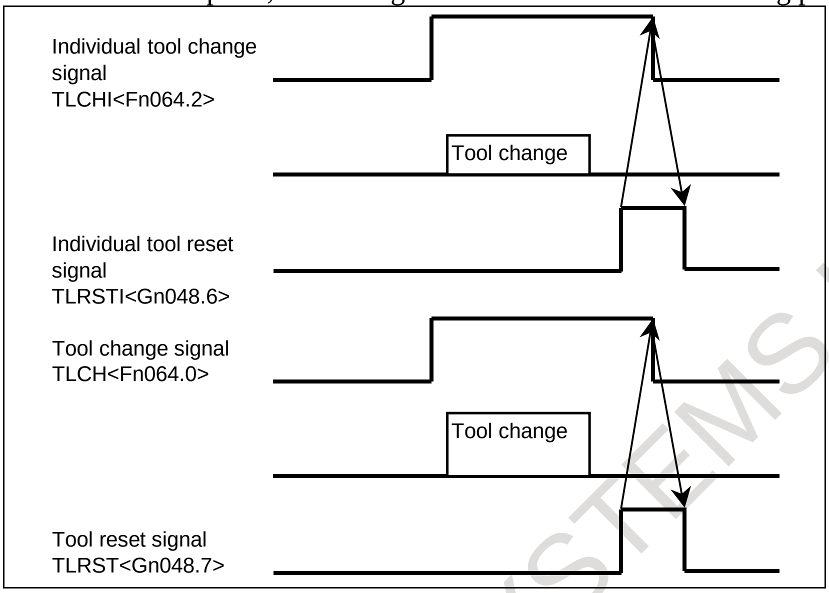
Data No.	Tool type No. (T)	Tool life counter	Maximum tool life value	Tool life status	Tool information	Tool length compensation No. (H)	Cutter compensation No. (D)
1	11111111	3300	3600	Not expired	UNTR	1	1

- N200 G01 X100.0 F100.0 ;

For the type for counting time, the time (in seconds) taken to execute such a cutting block is added to the counter value.

(1) The life count reaches 3600 seconds (the maximum tool life), and the tool life expires.
(2) The CNC changes the tool life status for tool management data number 1 to 3 (life has expired).
(3) The CNC searches for a tool that has the same tool type number and has a remaining life.
(4) Individual tool change signal TLCHI <Fn064.2> is set to "1".

(5) When the lives of all tools having the same tool type number have expired, tool change signal TLCH <Fn064.0> is set to “1“.
(6) Even when the tool life has expired, machining is continued until the machining program ends.



NOTE

Time is counted up while a cutting feed operation after the execution of M06 or a restart M code or T code is in progress. When the spindle is stopped or is rotating, the count-up operation is not performed. The target tools are the tools attached at the first to fourth spindle positions. When one of the following conditions is met, the count-up operation is not performed:

1 The tool life status indicates the invalid state (0) or breakage (4).
2 Even when a cutting feed is performed, the cutting feed is not regarded as being in progress.
   - When the system is in the FIN wait state
   - During in-position check
   - When the cutting feed override is 0%
   - When an interlock is provided
   - When the system is in the spindle speed arrival wait state

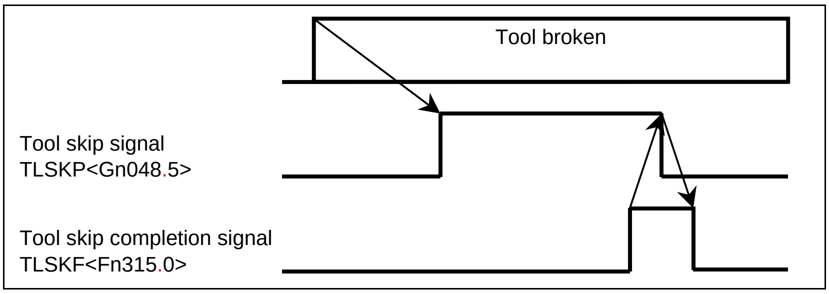
   (The above states can be confirmed from diagnosis data Nos. 000 to 013.)
   - When a machine lock is provided

- When the tool being used is broken

A tool breakage is detected by the machine and is posted to the CNC via the PMC. The PMC posts the tool breakage to the CNC in one of the following three methods:

- The tool life status is changed to 4 (tool breakage) by using the PMC window (function code 332)
- The tool life status is changed to 4 (tool breakage) by using the PMC window (function code 335).
- Tool skip signal TLSKP <Gn048.5> is set to “1“.

The tool placed in the tool breakage state is excluded from the target tools for tool life management when the next and subsequent T codes are specified.



- T code command for specifying a particular tool

To specify a particular tool directly without allowing the CNC to select a tool, use the following format:

M_ T_ ;

> **NOTE**
> A block for specifying the above command must not contain any other command.

M_ M code set in parameter No. 13252
T_
  If bit 0 (TCF) of parameter No. 13200 is 0:
    Upper 4 digits: Cartridge number (output to <Fn026 to Fn027>)
    Lower 4 digits: Pot number (output to <Fn028 to Fn029>)

If the T code command is not longer than 4 digits, it is output to <Fn026 to Fn027> of the PMC on the assumption that the cartridge number is 1.
To specify a spindle position or standby position, set the pot number to 0.

  Upper 4 digits: Cartridge number (11, 21)
  Lower 4 digits: 0001

If bit 0 (TCF) of parameter No. 13200 is 1:
  Any number from 0 to 99,999,999

If bit 0 (TCF) of parameter No. 13200 is 1 (outputting a T code directly to the output signal), the binary value of the specified T code is output to <Fn026 to Fn029> without modification.

> **NOTE**
> Pot numbers must not exceed 9999.

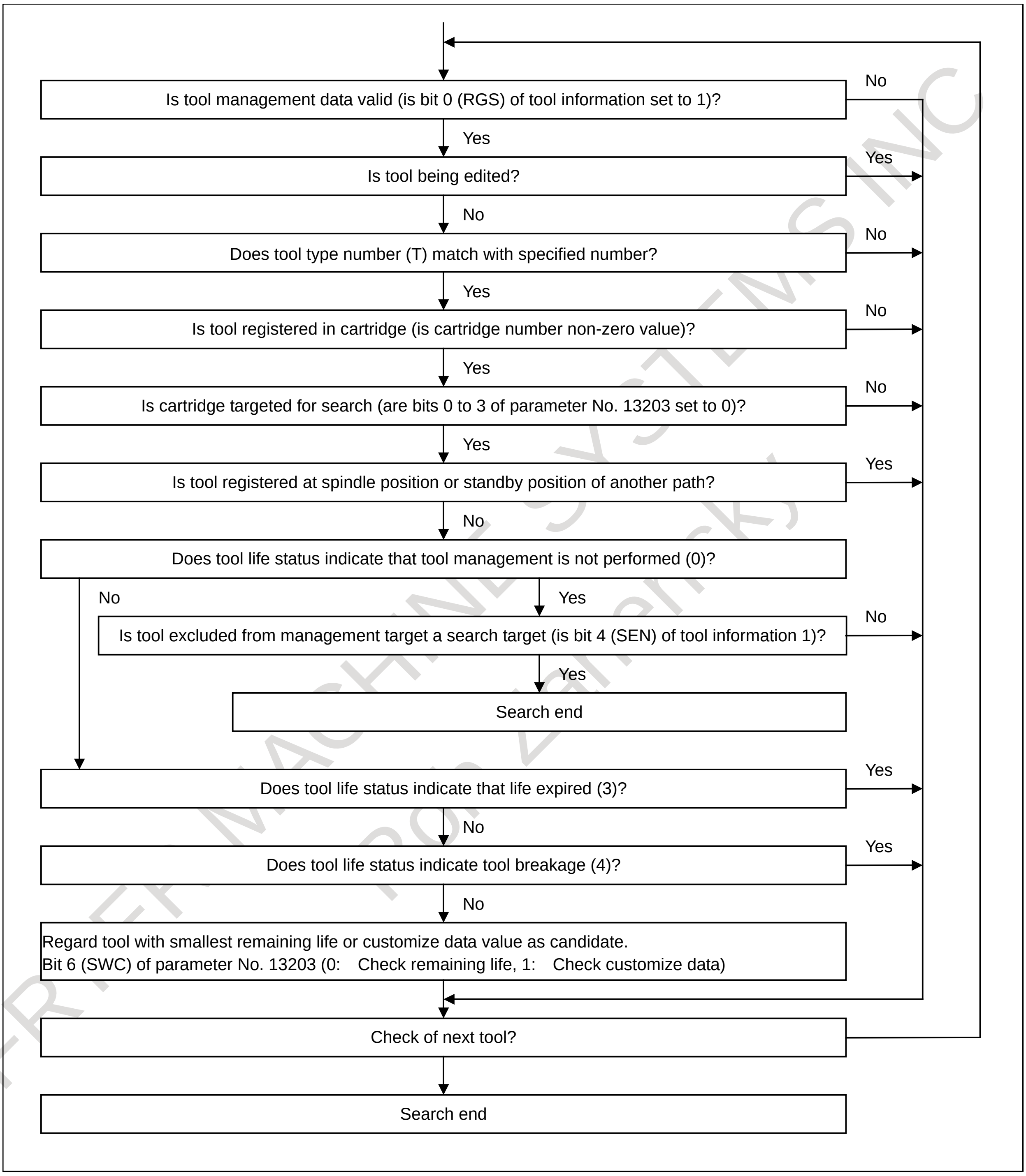
- Search by customize data

If bit 6 (SWC) of parameter No. 13203 is 0, a tool with the shortest life is searched for. If SWC is 1, a tool whose customize data with an arbitrary number holds the smallest is searched for. When SWC is set to 1, the number of the customize data by which a search operation is to be made is set in parameter No. 13260. When 3 is set in parameter No. 13260, the registered tools are searched to find a tool whose customize data 3 holds the smallest. If 0 is set in parameter No. 13260, a tool with the shortest life is searched for.

The customize data by which a search operation is to be made ranges from 1 to 99,999,999. If a negative value is set, its absolute value is assumed. If 0 is set, the tool is regarded as an invalid tool and is excluded from the tools to be searched.

- Tool search order

Tools having a tool type number (T) specified by a program are searched sequentially from tool management data number 1 while registered data contents are checked. The following shows how a search operation is made within the NC:



- System variables

The following tool management data (Table 11.3.1 (c)) of the tool being used as a spindle after a tool change by M06 and the tool to be used next which is specified by a T code can be read through custom macro variables:

Table 11.3.1 (c)

Being used	Item	
#8401	Data number	
#8402	Tool type number (T)	
#8403	Tool life counter	
#8404	Maximum tool life value	
#8405	Tool notice life value	
#8406	Tool life status	
#8407	Customize data 0 (bit)	
#8408	Tool information	
#8409	Tool length compensation number (H)	For machining center systems
#8410	Cutter compensation number (D)	
#8411	Spindle speed (S)	
#8412	Cutting feedrate (F)	
#8413	Tool geometry compensation number (G)	For lathe systems
#8414	Tool wear compensation number (W)	
#8431	Customize data 1	
#8432	Customize data 2	
#8433	Customize data 3	
#8434	Customize data 4	
#8435	Customize data 5	
#8436	Customize data 6	
#8437	Customize data 7	
#8438	Customize data 8	
#8439	Customize data 9	
#8440	Customize data 10	
#8441	Customize data 11	
#8442	Customize data 12	
#8443	Customize data 13	
#8444	Customize data 14	
#8445	Customize data 15	
#8446	Customize data 16	
#8447	Customize data 17	
#8448	Customize data 18	
#8449	Customize data 19	
#8450	Customize data 20	
#8451	Customize data 21	
#8452	Customize data 22	
#8453	Customize data 23	
#8454	Customize data 24	
#8455	Customize data 25	
#8456	Customize data 26	
#8457	Customize data 27	
#8458	Customize data 28	
#8459	Customize data 29	
#8460	Customize data 30	
#8461	Customize data 31	
#8462	Customize data 32	
#8463	Customize data 33	
#8464	Customize data 34	
#8465	Customize data 35	
#8466	Customize data 36	
#8467	Customize data 37	
#8468	Customize data 38	
#8469	Customize data 39	
#8470	Customize data 40	

When a cartridge number of a spindle position (11 to 14) or standby position (21 to 24) is specified in #8400, information about the corresponding position can be read.
If the spindle position table or standby position table has an empty pot, <empty> is read from #8402 to #8470.
Value 0 is read from #8401 (data number).
Therefore, machining conditions registered in tool management data can be specified directly by coding, for example, D#8410, H#8409, S#8411, and F#8412 with a tool change macro (such as M06). Similarly, customize data can be referenced by a custom macro, and machining programs can be customized according to the tool used.

- Specifying a tool compensation number

M

When parameter No. 13265 is 0, a compensation number registered as tool management data of a tool attached at a spindle position can be selected by specifying H99 or D99.
(99 is treated as a special number, so 99 cannot be specified directly as a compensation number.)
When other than 0 is set in parameter No. 13265, the number set in the parameter can be specified instead of 99. For example, if parameter No. 13265 is 3, specifying H3 specifies the tool length compensation number registered for the tool at the spindle position.

T

When the tool management function is not used, a tool compensation number is specified also with a T code; when the tool management function is used, the T code is used only to specify a tool type number, so a tool compensation number must be specified with address D.
Except the specifications for addresses, the specifications for the number of permissible digits (parameter No. 3032) and the number of digits consisting of a compensation number (parameter No. 5028), separation of geometry compensation numbers and wear compensation numbers (bit 1 (LGN) of parameter No. 5002), and so on are the same as for T.

If parameter No. 13265 is set to 0, when a compensation number registered for a tool attached at a spindle position is specified, the specification format varies according to the number of digits as follows, which is the same as for conventional T:
When the compensation number is 1 digit long: D9
When the compensation number is 2 digits long: D99
When the compensation number is 3 digits long: D999
Because 9, 99, or 999 is treated as a special number, it cannot be specified directly as a compensation number.
Unless parameter No. 13265 is set to 0, the number set in the parameter can be specified instead of 9, 99, or 999. When parameter No. 13265 is set to 3, specifying D3 specifies the tool geometry compensation number and tool wear compensation number registered for a tool attached at the spindle position.

Multi-path system

Depending on whether the local path is a machining center system or a lathe system, tool compensation numbers are specified by using one of the above methods.

Spindle selection

When specifying compensation numbers of a tool attached to a spindle other than the first spindle, specify the spindle number with address P within the same block that contains H/D. When specifying the first spindle, you can omit P.

D99 P3; Specifying compensation numbers registered for the tool attached at the third spindle
D99 ; Specifying compensation numbers registered for the tool attached at the first spindle

- Read/write operations for tool management data and cartridge management table

The tool management data and cartridge management table can be read from and written to by using the CNC/PMC data window library (FOCAS2). Therefore, a specific tool management system including all available tool data not registered in the CNC can be built easily by using the OPEN CNC.

Similarly, the tool management data and cartridge management table can also be read from and written to using the PMC window.

Tool management data of the tool being used as a spindle after a tool change operation performed by M06 and the tool being selected by specifying a T code can be read using a custom macro.

The user can modify tool management data by MDI from the tool management function screen of the CNC. Addition, modification, and deletion of the above tool management data can be made from a part program (G10). Such data can also be input to and output from external I/O equipment by using the ALL I/O screen and tool management function screen.

- G10 format

Addition, modification, and deletion are performed for the tool management data and cartridge management table from programs.

If a format error is found in the commands from G10 L75/L76/L77 to G11, or if a value beyond the valid data range is specified, alarm PS5312," ILLEGAL COMMAND IN G10 L75/76/77" is issued. In such a case, correct the program. Within the range from G10 to G11, no decimal point can be specified with any address. If a decimal point is specified, alarm PS0007," ILLEGAL USE OFDECIMAL POINT" results.

- Input to and output from external equipment

Input to and output from external equipment are enabled only on the ALL IO screen or tool management screen if the I/O unit number is 4, 6, or 7.

Caution

- The total number of pots of all cartridges is normally 64. It can be expanded to 240 and 1000 optionally.
- When pots are assigned to cartridges, the first cartridge has a top priority.

Example 1

When the maximum number of pots is 64

	Parameter setting	Number of available pots
First cartridge	30	30
Second cartridge	20	20
Third cartridge	20	14
Fourth cartridge	20	0

- If the content of one of parameter Nos. 13220, and 13222 to 13251 is rewritten, tool management data numbers of all cartridges are initialized to 0. This prevents conflict between tool management data and the cartridge management table.

Example 2

If parameter No. 13220 is set to 30 when tool management data number 60 is set for pot number 1 of cartridge 1, nonexistent tool data is regarded as being held in the pot.

- When data is edited, its tool management data number blinks, and it is excluded from the data to be searched for the next tool. In this case, you cannot modify the data by using the PMC or FOCAS2. This condition is maintained until the edit mode ends.

- When a name is set for customize data or the tool life status by using G10, a change to the set name is not made until the display screen is changed to another NC screen.

Signal

Tool change signal TLCH<Fn064.0>,TLCH1 to 4<Fn328.0 to Fn328.3>

[Classification] Output signal

[Function] These signals post that the life of the last one of the tools having the same tool type number has expired.

[Output cond.] These signals turn to “1“ in the following cases:

- When the life of the last one of the tools having the same tool type number has expired
- When the last one of the tools having the same tool type number is treated as a broken tool by tool skip signal TLSKP

These signals turn to “0“ in the following cases:

- When the tool change reset signal is set to “1“.

> **NOTE**
> 1 This signal is provided for each spindle position.
> 2 Fn064.0 is output signals for the first spindle.

Tool change reset signal TLRST<Gn048.7>,TLRST1 to 4<Gn328.0 to Gn328.3>

[Classification] Input signal

[Function] These signals set the tool change signal to “0“.

[Output cond.] When the signals are set to 1, the control unit operates as follows:

- Setting the tool change signal to “0“

> **NOTE**
> 1 Tool change signal TLCH is not cleared by reset.
> 2 This signal is provided for each spindle position.
> 3 Gn048.7 is input signals for the first spindle.

Individual tool change signals TLCHI<Fn064.2>,TLCHI1 to 4<Fn328.4 to Fn328.7>

[Classification] Output signal

[Function] Reports the end of the life of the current tool.

[Output cond.] These signals turn to “1“ in the following cases:

- The end of the life of the current tool is detected.

These signals turn to “0“ in the following cases:

- Individual tool-change reset is executed.

> **NOTE**
> 1 This signal is provided for each spindle position.
> 2 Fn064.2 is output signals for the first spindle.

Individual tool change reset signals TLRSTI<Gn048.6>,TLRSI1 to 4<Gn328.4 to Gn328.7>

[Classification] Input signal

[Function] Sets the individual tool change signal TLCHI to “0“.

[Operation] When the signals are set to “1“, the control unit operates as follows:

- Sets the individual tool change signal to “0“.

NOTE
1 Tool change signal TLCH is not cleared by reset.
2 This signal is provided for each spindle position.
3 Gn048.6 is input signals for the first spindle.

Tool skip signals TLSKP<Gn048.5>,TLSKP1 to 4<Gn329.0 to Gn329.3>

[Classification] Input signal

[Function] These signals can forcibly change a tool whose life has not yet expired. These signals are used when a tool is broken.

[Operation] When the signals are set to “1“, the control unit operates as follows:

- The tool life status of the tool currently used is set to 4 (tool breakage), and the tool is excluded from the tool life management targets.
- When there is no tool that has the same tool type number as the tool currently used and has a remaining life, tool change signal TLCH <Fn064.0> is set to “1“.
- Upon completion of a tool skip operation, tool skip completion signal TLSKF <Fn315.0> is set to “1“.
- The next T code command selects the next tool.

NOTE
1 This signal is provided for each spindle position.
2 Gn048.5 is input signals for the first spindle.

Tool skip completion signals TLSKF<Fn315.0>,TLSKF1 to 4<Fn329.0 to Fn329.3>

[Classification] Output signal

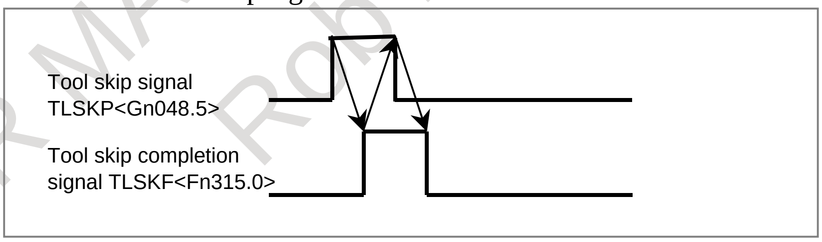
[Function] These signals post that the tool skip signal has been accepted.

[Output cond.] These signals turn to “1“ in the following cases:

- When tool skip signal TLSKP is set to “1“, and tool skip processing is completed When a tool not targeted for tool life management is being used, the tool skip processing is not performed, but the tool skip completion signal is set to “1“.

These signals turn to “0“ in the following cases:

- When the tool skip signal is set to “0“



NOTE
1 This signal is provided for each spindle position.
2 Fn315.0 is output signals for the first spindle.

Tool life count override signals *TLV0 to *TLV9<Gn049.0 to Gn050.1>

[Classification] Input signal

[Function] Overrides the life count (time) if bit 2 (LFV) of parameter No. 6801 is specified.
Each of the ten binary code signals has a unique override value that becomes valid when the signal is set to “0“. The life count is overridden by the sum of the valid override values. The override value can be specified in steps of 0.1, within the range of 0 to 102.3.

$$\text{Override value} = \sum_{i=0}^{9} \left\{ 2^i \times Vi \right\} \text{ times}$$

*TLV0	0.1	*TLV2	0.4	*TLV4	1.6	*TLV6	6.4	*TLV8	25.6
*TLV1	0.2	*TLV3	0.8	*TLV5	3.2	*TLV7	12.8	*TLV9	51.2

Example)

When *TLV7, *TLV6, and *TLV3 are set to “0“, the override value is calculated as follows:

12.8+6.4+0.8=20.0

The life count is multiplied by 20.0.

[Operation] The actual cutting time is counted and multiplied by the override value obtained by the signals. The calculated time is used as the basis for tool-life management.

Tool search in-progress signal TLMSRH<Fn315.1>

[Classification] Output signal

[Function] This signal posts that the CNC is searching for a tool.

[Output cond.] This signal turns to “1“ in the following cases:

- When the CNC is searching for a tool having the tool type number specified by a T code
- When the CNC is searching for a tool having the same tool type number as for the tool currently used whose life has expired during life management
- When tool skip signal TLSKP <Gn048.5> is input while the CNC is searching for a tool having the same tool type number

This signal turns to “0“ in the following cases:

- When a search operation is completed

> **NOTE**
> When tool search in-progress signal TLMSRH is 1, be careful not to rewrite tool management data by using the PMC window and FOCAS2.

Tool management data modification in-progress signal TLMG10<Fn315.2>

[Classification] Output signal

[Function] This signal posts that a modification to the tool management data, cartridge management table, or string data is being made by the G10 L75/L76/L77 command.

[Output cond.] This signal turns to “1“ in the following cases:

- When a modification to the tool management data or cartridge management table is being made by the G10 L75/L76/L77 command

This signal turns to “0“ in the following cases:

- When the modification mode of the tool management data and cartridge management table is ended by a G11 command, a reset, and so on

> **NOTE**
> When tool management data modification in-progress signal TLMG10 is 1, be careful not to rewrite tool management data by using the PMC window and FOCAS2.

Tool management data output in-progress signal TLMOT<Fn315.4>

[Classification] Output signal

[Function] This signal posts that the tool management data, cartridge management table, or string data is being output.

[Output cond.] This signal turns to “1“ in the following cases:

- When outputting of the tool management data or cartridge management table has started

This signal turns to 0 in the following cases:
- When outputting processing is completed

> **NOTE**
> When tool management data output in-progress signal TLMOT is 1, be careful not to rewrite tool management data by using the PMC window and FOCAS2.

Tool management data edit in-progress signal TLMEM<Fn315.7>

[Classification] Output signal

[Function] This signal posts that the tool management data edit mode is set.

[Output cond.] This signal turns to “1“ in the following cases:
- When the tool management data edit mode starts on the tool management data screen

This signal turns to “0“ in the following cases:
- When the tool management data edit mode ends on the tool management data screen

Tool life expiration notice signals TLCHB<Fn064.3>, TLCHB1 to 4<Fn329.4 to Fn329.7>

[Classification] Output signal

[Function] These signals post that the life of the tool being used has almost expired.

[Output cond.] These signals turn to “1“ in the following cases:
- When the life value of the currently used tool becomes the notice life value in the tool management data or less

  If bit 3 (ETE) of parameter No. 13200 is 0, these signals are output when the lives of all the tools having the same tool type number as the currently used tool have expired, and the currently used tool is the last tool. In this case, the same notice life value must be set for all tools having the same type number. (Previous notice of expiration for each type number)

  If bit 3 (ETE) of parameter No. 13200 is 1, these signals are output by checking only the life value of the currently used tool even when the life of a tool having the same tool type number as the currently used tool is still left. (Previous notice of expiration for each tool)

  Actually, these signals are set to “1“ when completion signal FIN is posted in response to an M06 command if the number of use times is specified; if time is specified, these signals are set to 1 during machining.

These signals turn to “0“ in the following cases:
- When the life value of the currently used tool becomes greater than the notice life value in the tool management data.
  Actually, these signals are set to “0“ when a change to a tool having a life value greater than the notice life value has been made.
- If the notice life value is set to 0, these signals are not output.

> **NOTE**
> Fn064.3 is input signals for the first spindle.

Tool life counting disable signals TLNCT1 to 4<Gn329.4 to Gn329.7>

[Classification] Input signal

[Function] These signals specify not to perform tool life counting for each spindle.

[Operation] When the signals are set to “1“, the control unit operates as follows:

- The CNC does not perform tool life management for the tool attached at the corresponding spindle position (the first to fourth spindle).

Life expiration signal TMFNFD<Fn315.6>

[Classification] Output signal

[Function] This signal indicates whether a valid tool whose life still remains is left among the tools having the type number specified by a T code.

[Output cond.] This signal turns to “1“ in the following cases:

When a T code is specified, the tools having the type number specified by the T code include no valid tool whose life still remains.

> **NOTE**
> In this case, it is necessary for the PMC to stop the operation by issuing an external alarm and request a tool change. If the selected tool is used continuously with no alarm issued, the life of the tool is counted up. The tool (cartridge and pot) selected by the CNC at this time is a tool having the largest tool management data number among the tools having a specified tool type number.

This signal turns to “0“ in the following cases:

- When a T code is specified, the tools having the type number specified by the T code include a valid tool whose life still remains.

> **NOTE**
> This signal is valid when bit 6 (NAM) of parameter No. 13200 is set to 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn048	TLRST	TLRSTI	TLSKP					
Gn049	*TLV7	*TLV6	*TLV5	*TLV4	*TLV3	*TLV2	*TLV1	*TLV0
Gn050							*TLV9	*TLV8
Gn328	TLRSI4	TLRSI3	TLRSI2	TLRSI1	TLRST4	TLRST3	TLRST2	TLRST1
Gn329	TLNCT4	TLNCT3	TLNCT2	TLNCT1	TLSKP4	TLSKP3	TLSKP2	TLSKP1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn026	T07	T06	T05	T04	T03	T02	T01	T00
Fn027	T15	T14	T13	T12	T11	T10	T09	T08
Fn028	T23	T22	T21	T20	T19	T18	T17	T16
Fn029	T31	T30	T29	T28	T27	T26	T25	T24
Fn064					TLCHB	TLCHI	TLNW	TLCH
Fn315	TLMEM	TMFNFD		TLMOT		TLMG10	TLMSRH	TLSKF
Fn328	TLCHI4	TLCHI3	TLCHI2	TLCHI1	TLCH4	TLCH3	TLCH2	TLCH1
Fn329	TLCHB4	TLCHB3	TLCHB2	TLCHB1	TLSKF4	TLSKF3	TLSKF2	TLSKF1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3108						**PCT**		

[Input type] Parameter input
[Data type] Bit path

#2 PCT For modal T display on the program check screen:
0: A specified T value is displayed.
1: HD.T and NX.T are displayed.
Values displayed follow bit 1 (THN) of parameter No. 13200.

	#7	#6	#5	#4	#3	#2	#1	#0
6801						**LVF**		

[Input type] Parameter input
[Data type] Bit path

#2 LVF When the life of a tool is counted in terms of time with the tool management function, the tool life count override signals *TLV0 to *TLV9 <Gn049.0 to Gn050.1> are:
0: Invalid.
1: Valid.

> **NOTE**
> The use of this parameter varies depending on whether the tool management function or tool life management function is used.

6811	**Tool life count restart M code**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 127 (not including 01, 02, 30, 98, and 99)

When 0 is set, this parameter is ignored.
When an M code for tool life count restart is specified, the counting of the life of the tool attached at the spindle position is started.
When the type for counting the number of use times is selected, the target of life counting is switched to the tool attached at the spindle position, and the life count is incremented by 1.
When the type for counting time is selected, the target of life counting is switched to the tool attached at the spindle position, with no other operations performed.
If the tool attached at the spindle position is not a tool under tool life management, no operation is performed.

> **NOTE**
> The use of this parameter varies depending on whether the tool management function or tool life management function is used.

	#7	#6	#5	#4	#3	#2	#1	#0
13200	NFD	NAM	T0O	TP2	ETE	TRT	THN	TCF

[Input type] Parameter input
[Data type] Bit path

#0 TCF When a T code is specified with the tool management function:
0: A cartridge number and pot number found by the NC are output.
1: The specified T code is output without modification.

#1 THN When NX.T and HD.T are displayed with the tool management function:
0: The tool type numbers at the first spindle position and the first standby position are displayed.
1: The values specified from the PMC window are displayed.

#2 TRT As the remaining lifetime value for outputting the tool life arrival notice signal:
0: The remaining lifetime of the last tool is used.
1: The sum of the remaining lifetimes of the tools with the same type number is used.

NOTE
This parameter is valid when bit 3 (ETE) of parameter No. 13200 is set to 0 (arrival notice for each type number).

#3 ETE The tool life arrival notice signal is output:
0: For each tool type.
1: For each tool.

#4 TP2 The output format of cartridge management data is:
0: New registration format (G10L76P1 format).
1: Modification format (G10L76P2 format).

#5 T0O When T0 is specified:
0: A tool search is made assuming that the tool type number is 0.
1: The cartridge number and pot number are assumed to be 0.

#6 NAM When a T code is specified, but a valid tool with a remaining lifetime cannot be found:
0: The alarm PS5317, "LIVES OF ALL TOOLS EXPIRED" is issued.
1: The alarm is not issued. Instead, the tool with the maximum tool management number is selected from the tools of the specified tool type, and TMFNFD<F315.6> is set to “1“.

#7 NFD When a T code is specified, but a valid tool with a remaining lifetime cannot be found in the cartridge:
0: The spindle position and standby position are also searched.
1: The spindle position and standby position are not searched.

	#7	#6	#5	#4	#3	#2	#1	#0
13201							TDN	

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 TDN On the tool management function screen, the character string for indicating the tool life status can contain:
0: Up to 6 characters.
1: Up to 12 characters.

	#7	#6	#5	#4	#3	#2	#1	#0
13202	DOM	DOT		DO2		DOY	DCR	

[Input type] Parameter input
[Data type] Bit

#1 DCR On the tool management function screen, tool nose radius compensation data is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the lathe system or compound system.

#2 DOY On the tool management function screen, Y-axis offset data is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the lathe system or compound system.

#4 DO2 On the tool management function screen, the second geometry tool offset data is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the lathe system or compound system.

#6 DOT On the tool management function screen, the tool offset data (X, Z) of the T series is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the lathe system or compound system.

#7 DOM On the tool management function screen, the tool offset data of the M series is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the machining center system or compound system.

	#7	#6	#5	#4	#3	#2	#1	#0
13203	TCN	SWC			NM4	NM3	NM2	NM1

[Input type] Parameter input
[Data type] Bit path

#0 NM1 The first cartridge is:
0: Searched.
1: Not searched.

#1 NM2 The second cartridge is:
0: Searched.
1: Not searched.

#2 NM3 The third cartridge is:
0: Searched.
1: Not searched.

#3 NM4 The fourth cartridge is:
0: Searched.
1: Not searched.

#6 SWC The tools with the same tool type number are searched for:
0: Tool with the shortest lifetime.
1: Tool with the small customization data number.
In this case, a customization data number is to be set in parameter No. 13260.

#7 TCN Tool life count operation is triggered by:
0: M06/restart M code. (A T code alone does not start counting.)
1: T code. (Count operation is not started by M06.)

13220	Number of valid tools in tool management data

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 64 (Extended to 240 or 1000 by the addition of an option)
This parameter sets the number of valid tools in tool management data.

13221	M code for tool life count restart

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 65535
When 0 is set in this parameter, this parameter is ignored.

When an M code for tool life count restart is specified, the counting of the life of the tool attached at the spindle position is started. When the type for counting the number of use times is selected, the target of life counting is switched to the tool attached at the spindle position, and the life count is incremented by 1.
When the type for counting time is selected, the target of life counting is switched to the tool attached at the spindle position but no other operations are performed. If the tool attached at the spindle position is not a tool under tool life management, no operation is performed.
The M code set in parameter No. 6811 waits for FIN. However, the M code set in this parameter does not wait for FIN.
The M code set in parameter No. 13221 must not be specified in a block where another auxiliary function is specified.
The M code set in parameter No. 13221 does not wait for FIN. So, do not use the M code for other purposes.

NOTE
The use of this parameter varies depending on whether it is used by the tool management function or tool life management function.

13222	Number of data items in the first cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 64 (Extended to 240 or 1000 by the addition of an option)
This parameter sets the number of data items used with the first cartridge.

13223	Start pot number of the first cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 9999
This parameter sets the start pot number to be used with the first cartridge. Pot numbers starting with the value set in this parameter and sequentially incremented by 1 are assigned to all data items.

13227	Number of data items in the second cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 64(Extended to 240 or 1000 by the addition of an option)
This parameter sets the number of data items used with the second cartridge.

13228	Start pot number of the second cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 9999

This parameter sets the start pot number to be used with the second cartridge. Pot numbers starting with the value set in this parameter and sequentially incremented by 1 are assigned to all data items.

13232	Number of data items in the third cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 64(Extended to 240 or 1000 by the addition of an option)

This parameter sets the number of data items used with the third cartridge.

13233	Start pot number of the third cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 9999

This parameter sets the start pot number to be used with the third cartridge. Pot numbers starting with the value set in this parameter and sequentially incremented by 1 are assigned to all data items.

13237	Number of data items in the fourth cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 64(Extended to 240 or 1000 by the addition of an option)

This parameter sets the number of data items used with the fourth cartridge.

13238	Start pot number of the fourth cartridge

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 9999

This parameter sets the start pot number to be used with the fourth cartridge. Pot numbers starting with the value set in this parameter and sequentially incremented by 1 are assigned to all data items.

13250	Number of valid spindles

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 4

This parameter sets the number of spindle positions usable with the tool management function.

13251	Number of valid standby positions

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 4

This parameter sets the number of standby positions usable with the tool management function.

13252	M code for specifying a particular tool

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 65535

This parameter sets not a tool type number but an M code for directly specifying the T code of a particular tool.

13260	Customization data number to be searched for

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 40

When bit 6 (SWC) of parameter No. 13203 is set to 1, this parameter sets a customization data number to be searched for.

The valid data range is 1 to 4 when the option for customization data extension is not selected. When the option for customization data extension (5 to 20) is selected, the valid data range is 1 to 20. When the option for customization data extension (5 to 40) is selected, the valid data range is 1 to 40.
When bit 6 (SWC) of parameter No. 13203 is set to 0, or a value not within the valid data range is set, the search function based on customization data is disabled, and the tool with the shortest lifetime is searched for.

13265	Number for selecting a spindle position offset number

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 9999

This parameters sets an H/D code for selecting an offset number registered in the data of the tool attached at the spindle position.
When 0 is set, an ordinary used code such as H99/D99 is used. When a value other than 0 is set, H99/D99 no longer has a particular meaning. So, when H99/D99 is specified in this case, the specification of offset number 99 is assumed.
With the T series, address D only is used to specify a tool number and offset number, so that a restriction is imposed on the number of digits. So, the valid data range of this parameter varies according the number of digits of an offset number.
When the number of digits of an offset number is 1: to 9
When the number of digits of an offset number is 2: to 99
When the number of digits of an offset number is 3: to 999
When parameter No.5028 is set, the data range is as follows regardless of the number of digits of an offset number.
When No.5028 is set to 1: to 9
When No.5028 is set to 2: to 99
When No.5028 is set to 3: to 999
When No.5028 is set to 4: to 9999

NOTE
The use of this parameter varies depending on whether it is used by the tool management function or tool life management function.

Alarm and message

Number	Message	Description
PS0154	NOT USING TOOL IN LIFE GROUP	H99 or D99 is specified when no tool management data number is assigned to the spindle position. Correct the program.
PS0374	ILLEGAL REGISTRATION OF TOOL MANAGER(G10)	G10L75 or G10L76 data was registered during the following data registration: - From the PMC window. - From the FOCAS2. - By G10L75 or G10L76 in another system. Command G10L75 or G10L76 again after the above operation is completed.
PS5312	ILLEGAL COMMAND IN G10 L75/76/77	One of formats in G10L75, G10L76, or G10L77 to G11 commands is in error, or the command value is out of data range. Modify the program.
PS5316	TOOL TYPE NUMBER NOT FOUND	A tool with the specified tool-type number could not be found. Modify the program or register the tool.
PS5317	ALL TOOL LIFE IS OVER	The lives of all tools with the specified tool-type number have expired. Replace the tool.

Limitation

When this function is enabled, in lathe systems an offset number is specified with address D, so D cannot be used for other purposes. As arguments of macro calls in custom macros, D can be specified as done before.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Tool management function

11.3.2 Tool Management Extension Function

The following functions have been added to the tool management function:
1. Customization of tool management data display
2. Setting of spindle position/standby position display
3. Input of customize data with the decimal point
4. Protection of various tool information items with the KEY signal
5. Selection of a tool life count period
6. Each tool data screen
7. Total life time display for tools of the same type

They will be explained in detail below.

11.3.2.1 Customization of tool management data display

With the tool management data screen display customization function, the display positions of screen elements (type number, tool information, life counter, and so forth) on the tool management screen can be changed and whether to display or hide such screen elements can be chosen using the G10 format. This function enables a customized tool management screen to be configured.

11.3.2.2 Setting of spindle position/standby position display

In MG on the tool management data screen, a spindle position or standby position is displayed as a number such as 11, 12, and 13. With the spindle position/standby position display setting function, three arbitrary characters can be displayed using the G10 format.

11.3.2.3 Input of customize data with the decimal point

With the function for input of customize data with the decimal point, the number of decimal places can be set using the G10 format for each customize data item (customize data 1, ..., 40) to enable data input with the decimal point.

11.3.2.4 Protection of various tool information items with the KEY signal

When tool management data is in the edit state, various information items can be modified. By setting bit 0 (TDL) of parameter No. 13204 to 1, tool management data can be protected with the KEY signal so that various information items are not registered, modified, and deleted.

11.3.2.5 Selection of a tool 5life count period

A tool life count period can be chosen between 1 sec and 8 msec on a tool-by-tool basis.
Bit 5 of tool information is used to make a life count period selection.

Item	Description
Data length	1 byte (flag data)
#5 REV	0 : A life count period of 1 sec is used. (S) 1 : A life count period of 8 msec is used. (M)

Range of count is as follows.
1sec : 0 to 3,599,999 seconds (999 hours 59 minutes 59 seconds)
8msec: 0 to 3,599,992 ms (59 minutes 59 seconds 992 milliseconds)

NOTE
This function is valid when the tool information TIM (#1) is set to 1.

11.3.2.6 Each tool data screen

All data for a specified tool can be extracted and displayed.

11.3.2.7 Total life time display for tools of the same type

The remaining lives of tools with the same type numbers are totaled, and totals are displayed in order by tool type number or by remaining life. Also, tools with the same tool type numbers are displayed in a list.

- Total life data display

Total life data screen display changes in screen configuration depending on whether the tool life arrival notice signal is to be output for each tool type or for each tool.
The tool life arrival notice signal changes in output method depending on the settings of bit 3 (ETE) of parameter No. 13200 and bit 2 (TRT) of parameter No. 13200.
Under the following conditions, the tool life arrival notice signal is output for the same tool type number, and the total notice life and the state are added to the display items on the total life data screen.

- Details of data

Bit 3 (ETE) of parameter No. 13200	Bit 2 (TRT) of parameter No. 13200	Description of display
0	1	Total notice life and state items are displayed.

The details of the data displayed on the total life data screen are as follows:

- Sort No.

Item	Description
Data length	2byte
Valid data range	1 to maximum number of tools (64, 240, or 1,000)

The number representing a tool of a different type is displayed. The maximum number of tools differs with the options used.

- Tool type number (T code)

Item	Description
Data length	4byte
Valid data range	1 to 99,999,999

The tool type number (T code) set on the tool management data screen is displayed.

- Total remaining life

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times Time specifications: Seconds/milliseconds
Valid data range	When the number of use times is specified: 0 to2,147,483,647 times When time is specified: 0 to 359,999,999 s (99999 hours 59 minutes 59 seconds) 0 to 359,999,999 milliseconds (99 hours 59 minutes 59 seconds 999 milliseconds)

The remaining lives of tools with the same tool type number, as determined by subtracting the life counter value from the maximum life, are totaled for each number of times or for each time and displayed.

- Total life counter

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Time When time is specified: Seconds/milliseconds
Valid data range	When the number of use times is specified: 0 to 2,147,483,647 times When time is specified: 0 to 359,999,999 s (99999 hours 59 minutes 59 seconds) 0 to 359,999,999 milliseconds (99 hours 59 minutes 59 seconds 999 milliseconds)

The life counter values of the same-type tools set on the tool management data screen (for each tool type number) are totaled for each number of times or for each time and displayed.

- Total maximum life

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Time When time is specified: Seconds/milliseconds
Valid data range	When the number of use times is specified: 0 to 2,147,483,647 time When time is specified: 0 to 359,999,999 s (99999 hours 59 minutes 59 seconds) 0 to 359,999,999 milliseconds (99 hours 59 minutes 59 seconds 999 milliseconds)

The maximum lives of the same-type tools set on the tool management data screen (for each tool type number) are totaled for each number of times or for each time and displayed.

- Number (number of tools)

Item	Description
Data length	2byte
Valid data range	1 to maximum number of tools (64, 240, or 1,000)

The numbers of tools of the same types are totaled and displayed. The maximum number of tools change depending on the options used.

- Total notice life

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds/milliseconds
Valid data range	When the number of use times is specified: 0 to 2,147,483,647 times When time is specified: 0 to 359,999,999 s (99999 hours 59 minutes 59 seconds) 0 to 359,999,999 milliseconds (99 hours 59 minutes 59 seconds 999 milliseconds)

The notice lives of same-type tools set on the tool management data screen (for each tool type number) are totaled for each number of times or for each time and displayed.

- Status

Item	Description
Data length	1byte
Data details	0: -- 1: Notice

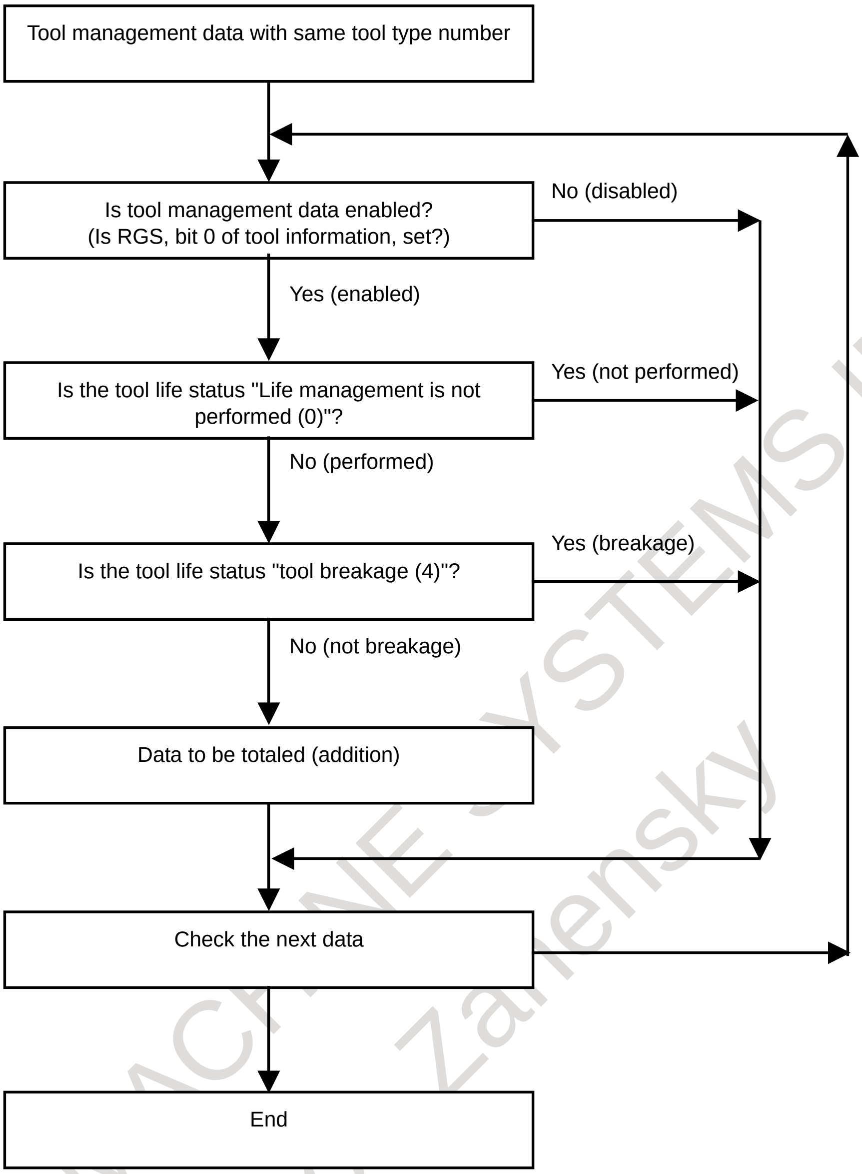
The status indicating whether the tool life arrival signal has been output is displayed.

- Sort function

On the total life data screen, sorting in either ascending or descending order is possible with the following data:

- Tool type number
- Total remaining life
- Data to be totaled

The data to be totaled on the total life data screen is that for those tools that satisfy the following conditions.



NOTE

1 If data is with the same tool type number and both time and number of times are specified as life count types, and the data is to be totaled, the data is totaled for both each time and each number of times and displayed in the time and number of times fields.
2 In the number column, the number of all tools with the same tool type number is displayed regardless of the tool life status.
3 The total life counter, total remaining live, total maximum life, and total notice life are displayed in time display in sec units if data in sec and in ms exist in the same tool. Any fractional part of ms data is truncated.
4 If the total remaining life is negative, zero is displayed.
5 If the total life counter, total remaining live, total maximum life, or total notice life is larger than the display maximum, an asterisk "*" is displayed.
Number of times:**********
Time:*****H**M**S (in sec. units)/ **H**M**S*** (in msec. units)

> **NOTE**
> 6 If one of the following operation is performed, total life data is updated:
> (a) Pressing of the soft key [UPDATE] or [SWITCH]
> (b) Pressing of the soft key [T-ASCE-SORT], [T-DESC-SORT], [R-ASCE-SORT], or [R-DESC-SORT]
> (c) Switching to another screen and return to the total life data screen

- Detailed life data display

The detailed life data screen displays the tool information specific to the same-type tools displayed on the total life data screen.

- Data details
  The details of the data displayed on the detailed life data screen are as follows:

- Tool type number (T code)

Item	Description
Data length	4byte
Valid data range	1 to 99,999,999

The tool type number (T code) set on the tool management data screen is displayed.

- Order

Item	Description
Data length	2byte
Valid data range	1 to maximum number of tools (64, 240, or 1,000)

Those tools whose tool life status is "enabled" are displayed in either of the following orders:
- If bit 6 (SWC) of parameter No. 13203 is 0, the order starting with the one with the shortest remaining life.
- If bit 6 (SWC) of parameter No. 13203 is 1, the order starting with the smallest customize data number set in parameter No. 13260.

For those tools whose tool life status is not "Tool not yet used (1)" or "Life remains (2)", the order item is displayed as an "-" (hyphen), and those tools are displayed in order by tool management data number, after ordered tools.
The maximum for the valid data range differs depending on the option used.

- Remaining tool life value

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds/milliseconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds) 0 to 3,599,999 milliseconds (59 minutes 59 seconds 999 milliseconds)

- The remaining life value is equal to [Maximum tool life value - Tool life counter value]. If the maximum tool life value is less than the tool life counter (resulting in a negative value), 0 is displayed.

- Tool life counter

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds/milliseconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds) 0 to 3,599,999 milliseconds (59 minutes 59 seconds 999 milliseconds)

- Maximum tool life value

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds/milliseconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds) 0 to 3,599,999 milliseconds (59 minutes 59 seconds 999 milliseconds)

- Notice life value

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds/milliseconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds) 0 to 3,599,999 milliseconds (59 minutes 59 seconds 999 milliseconds)

- Tool life status

Item	Description
Data length	1byte
Data details	0: Life management is not performed. 1: Tool not yet used 2: Life remains. 3: Life expired. 4: Tool breakage (skip)

NOTE

1 If, in a multi-path system, customize data number is specified for the order specification (bit 6 (SWC) of parameter No. 13203 =1), the order indicated is based on the customize data number specified in parameter No. 13260 of the path currently displayed.

2 Detailed life data is updated if either of the following operation is performed:
(a) Pressing of the soft key [UPDATE]
(b) Switching to another screen and return to the total life data details screen

Signal

Tool management data protection signals TKEY0 to TKEY5<G330.0 to G330.5>

[Classification] Input signal

[Function] These signals permit those operations from the MDI unit that will change the memory description. Six signals are available, as described below, which protect different tool information.

If TDL, bit 0 (TDL) of parameter No. 13204, is 0

- The tool management data protection signals are invalid.

If TDL is 1

- TKEY0 : Permits the input of tool type numbers.
- TKEY1 : Permits the input of tool information. (If tool management function oversize tool support is used, permits the input of tool shape numbers.)
- TKEY2 : Permits the input of the tool life counter value, maximum tool life value, notice life value, tool life status.
- TKEY3 : Permits the input of tool compensation-related information (such as tool length compensation number and cutter compensation number).
- TKEY4 : Permits the input of the spindle speed (S)/feedrate (F).
- TKEY5 : Permits the input of customize data-related data (customize data items 0, ..., 40).

[Operation] If a signal is 0, the corresponding operation is prohibited.
If a signal is 1, the corresponding operation is permitted.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G330			TKEY5	TKEY4	TKEY3	TKEY2	TKEY1	TKEY0

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
13200					ETE	TRT		

[Input type] Parameter input
[Data type] Bit path

#2 TRT As the remaining lifetime value for outputting the tool life arrival notice signal:
0: The remaining lifetime of the last tool is used.
1: The sum of the remaining lifetimes of the tools with the same type number is used.

> **NOTE**
> This parameter is valid when bit 3 (ETE) of parameter No. 13200 is set to 0 (arrival notice for each type number).

#3 ETE The tool life arrival notice signal is output:
0: For each tool type.
1: For each tool.

	#7	#6	#5	#4	#3	#2	#1	#0
13201						TDB		TDC

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 TDC The function of customizing the tool management data screen of the tool management function is:
0: Disabled.
1: Enabled.

#2 TDB The tool management function displays tool information in the:
0: Conventional mode.
1: 1/0 mode.

	#7	#6	#5	#4	#3	#2	#1	#0
13204								TDL

[Input type] Parameter input
[Data type] Bit

#0 TDL The protection function for tool management data using a key is:
0: Disabled.
1: Enabled.

Alarm and message

Number	Message	Description
PS5312	ILLEGAL COMMAND IN G10 L75/76/77	In commands from G10 L75/L76/L77 to G11, there is a format error, or a value beyond the specifiable range is specified. Correct the program.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Tool Management Extension Function

11.3.3 Tool Management Function Oversize Tools Support

Overview

Tool management function oversize tools support is added to the tool management function.
The figure of an oversize tool can be defined freely, and the figure of each oversize tool is registered. When an oversize tool is stored in a cartridge, interference with tools stored in other pots is considered. This function is usable with cartridges of chain type and matrix type.

Explanation

Usually, a tool, when stored in a cartridge, occupies one pot only. An oversize tool occupies pots other than the position where the tool is stored. This point is taken into consideration for tool management.
When the tool information #2 of the tool management data is set to 1, the tool is set as an oversize tool.
- Tool information

Item	Description
Data length	1byte (Flag data)
#2 BDT	0: Normal tool (N) 1: Oversize tool (B)

Oversize tools each having a different figure can be freely defined to form oversize tool patterns through MDI input on the tool figure setting screen, G10 data input, or inputting a file. Up to 20 patterns can be registered. An identification number (hereinafter referred to as a tool figure number) is assigned to each patterned figure, and a tool figure number suitable for each tool figure is set in the tool management data.

- Tool geometry number (GNO)

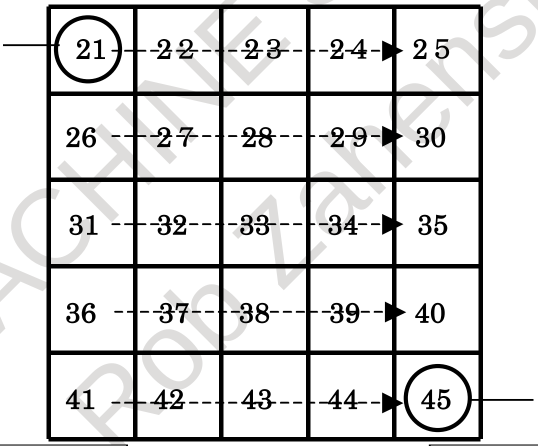
Item	Description
Data length	2byte
Valid data range	0 to 20

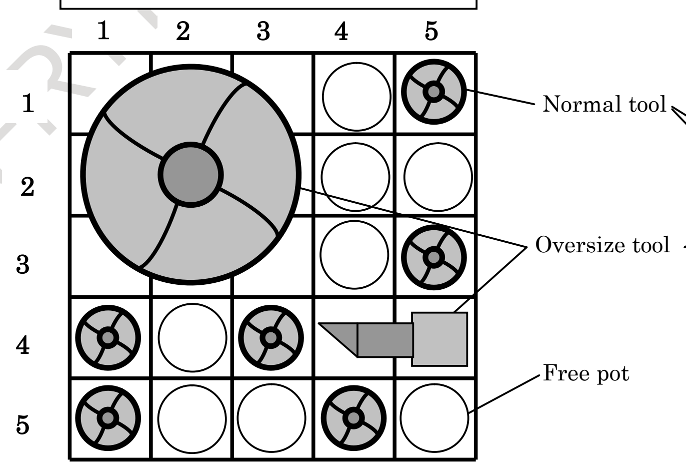
When a tool is registered in the cartridge management table, the figure of the tool is determined from the tool figure number of the tool so that the tool can be registered only if the tool does not interfere with other tools.

When using this function, set the type of cartridge with parameters. For the chain type, set a pot start number and the number of pots with the conventional parameters. When using the matrix type, use the same parameter as for the chain type to specify a pot start number, and use the parameters for specifying the number of pots in the vertical direction (hereinafter referred to as the number of rows) and the parameter for specifying the number of pots in the horizontal direction (hereinafter referred to as the number of columns) to specify the number of pots. As shown in the figure below, processing is performed, assuming that pot numbers are assigned from the upper-left corner to the lower-right corner of the cartridge as viewed when you face the front of the cartridge.

No tool can be stored in a way that causes interference with the outer frame of the cartridge.

When pattern start number is 21





Tool management table

Tool manage-ment No.	Tool information	Tool figure No.
1	0000UBCR	1
2	0000UNCR	0
3	0000UBCR	4
4	0000UNCR	0
5	0000UNCR	0
6	0000UNCR	0
7	0000UNCR	0
8	0000UNCR	0

Cartridge management table

Pot No. (Row-Column)	Tool management No.	Pot No. (Row-Column)	Tool management No.	Pot No. (Row-Column)	Tool management No.
1(01-01)	*	11(03-01)	*	21(05-01)	8
2(01-02)	*	12(03-02)	*	22(05-02)	0
3(01-03)	*	13(03-03)	*	23(05-03)	0
4(01-04)	0	14(03-04)	0	24(05-04)	4
5(01-05)	5	15(03-05)	6	25(05-05)	0
6(02-01)	*	16(04-01)	2		
7(02-02)	1	17(04-02)	0		
8(02-03)	*	18(04-03)	7		
9(02-04)	0	19(04-04)	*		
10(02-05)	0	20(04-05)	3		

Oversize tool

Pots occupied by oversize tool

Tool figure setting screen

Tool figure No.	Left direction	Right direction	Upper direction	Lower direction	Figure
1	2	2	2	2	A
2	1	1	1	1	A
3	2	3	3	1	A
4	2	0	0	0	A
5	2	2	2	2	A
Omitted					
19	1	3	2	2	A
20	2	2	4	4	A

NOTE

1 Even if a tool figure number is set in the tool management data, the tool is regarded as a normal tool when the tool information #2 is set to 0.

2 If all data items corresponding to a tool figure number are set to 0 (0 for the upper, lower, left, and right directions), the tool is regarded as a normal tool even when the tool information #2 is set to 1.

- Tool figure setting screen

The area occupied by a tool depends on whether the tool is stored in a cartridge of matrix type or a cartridge of chain type. A tool stored in a cartridge of matrix type occupies pots in the upper, lower, left, and right directions. A tool stored in a cartridge of chain type occupies pots only in the left and right directions.

- Cartridge of matrix type

The area occupied by a tool stored in a cartridge of matrix type is described below.
Set left, right, upper, and lower pots occupied relative to a reference pot in steps of 0.5 pot. The maximum specifiable value is 4 (for 2 pots). (See Fig. 11.3.3 (a).)
As shown in Fig. 11.3.3 (b), the pots located in the slant directions relative to a reference pot are automatically assumed to be occupied areas so that the occupied areas form a rectangle. However, if the same value, 3 or a greater value, is set for the left, right, upper, and lower directions, and figure B is selected, the four corners of the square can be excluded from the occupied areas as shown in Fig. 11.3.3 (c). When a tool has a circular figure, the tool can be stored efficiently in pots if figure B is selected.

Table 11.3.3 (a) Tool figure data

Tool pattern No.	Left direction	Right direction	Upper direction	Lower direction	Figure
1	0	3	2	1	A
2	2	2	1	3	A
Omitted					
19	1	1	1	1	A
20	2	3	2	2	A

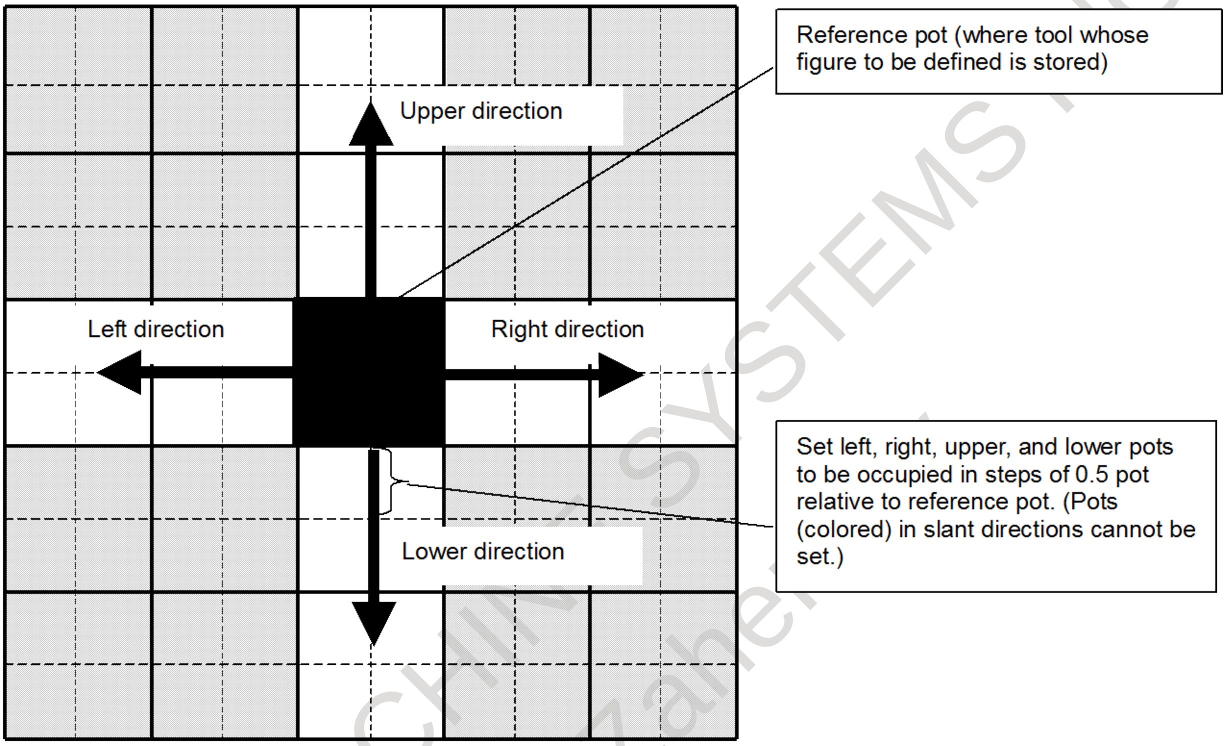


Fig. 11.3.3 (a) Tool figure data setting

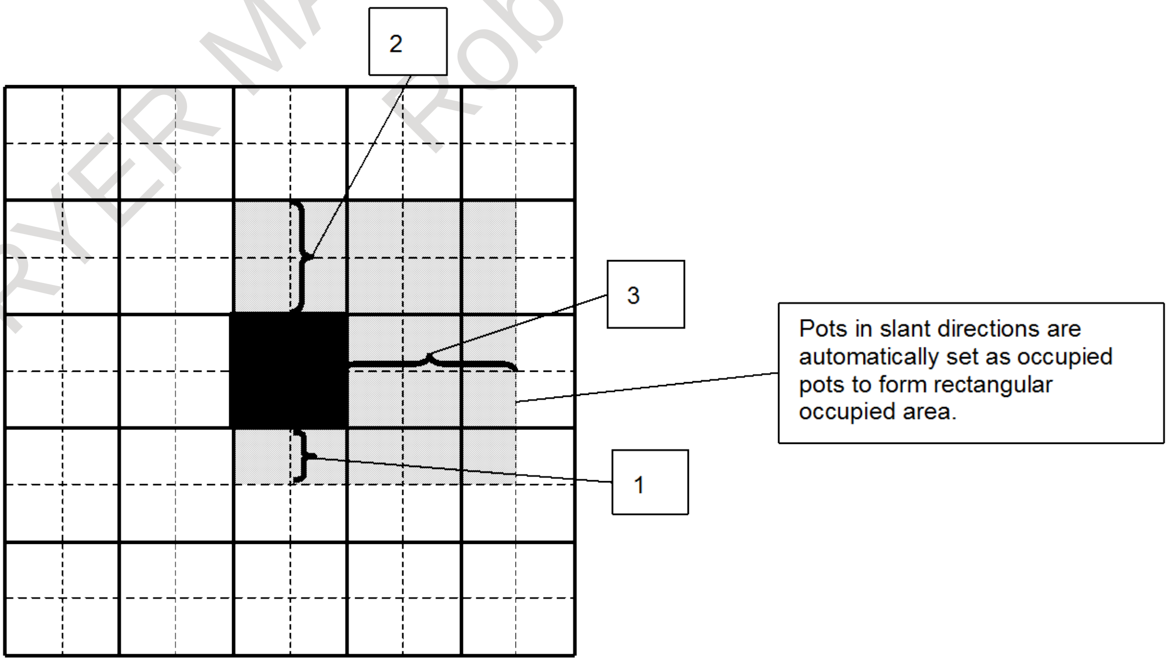


Fig. 11.3.3 (b) For tool figure pattern number 1 in Table 11.3.3 (a)

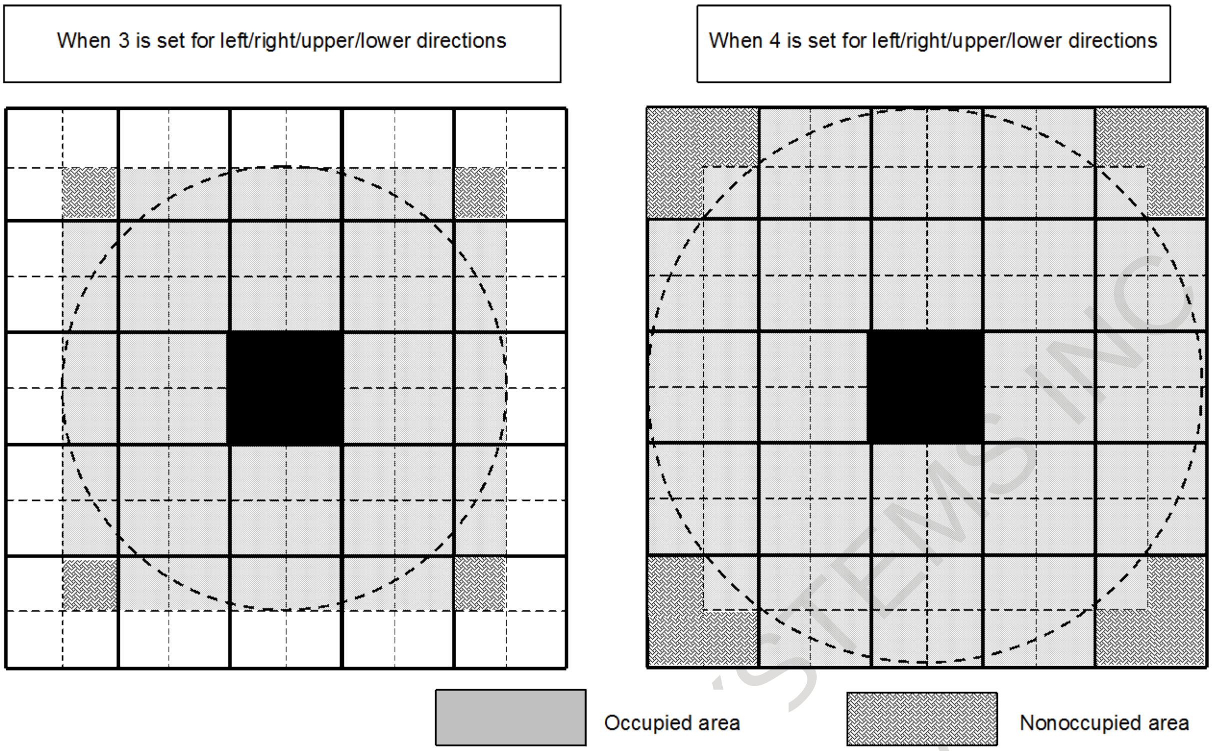


Fig. 11.3.3 (c) Occupied area for figure B

- Cartridge of chain type

The area occupied by a tool stored in a cartridge of chain type is described below.
As with the matrix type, input data on the tool figure setting screen. In this case, set pots to be occupied only in the left and right directions. The specification of pots in the upper and lower directions and a figure are ignored.

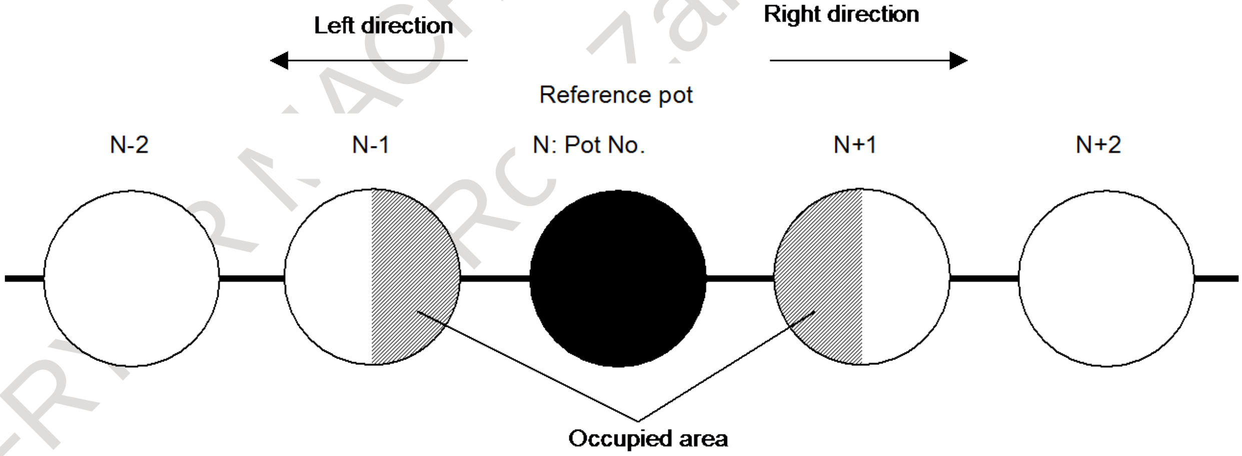


Fig. 11.3.3 (d) For tool figure data 19

NOTE

1. Tool figure data cannot be modified if the tool for which the tool figure number is set is registered in the cartridge.
2. When changing the tool figure number of a tool registered in the cartridge, ensure that the figure after the change does not interfere with other tools. If interference with other tools occurs, a warning is issued to disable an attempt to change the tool figure number.

- System variables

The following tool management data of the tool being used as a spindle after a tool change by M06 and the tool to be used next which is specified by a T code can be read through custom macro variables. The custom macro variable is a read-only variable.

Number	Item
#8415	Tool figure number (A)

- Tool management extension function

- Customization of tool management data display
  In customization of tool management data display with the tool management extension function, tool figure number display can be customized.

R	Item	Display width
12	Tool figure number (A)	3

- Protection of diverse tool information with KEY signals
  With the tool management data protection signal, TKEY1 <G330.1>, the input of tool shape numbers using MDI keys is prohibited.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
13240					MT4	MT3	MT2	MT1

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 MT1 The first cartridge is of the:
0: Chain type.
1: Matrix type.
When this parameter is set to 1, parameter No. 13222 is invalid.

#1 MT2 The second cartridge is of the:
0: Chain type.
1: Matrix type.
When this parameter is set to 1, parameter No. 13227 is invalid.

#2 MT3 The third cartridge is of the:
0: Chain type.
1: Matrix type.
When this parameter is set to 1, parameter No. 13232 is invalid.

#3 MT4 The fourth cartridge is of the:
0: Chain type.
1: Matrix type.
When this parameter is set to 1, parameter No. 13237 is invalid.

13241	Number of rows of the first cartridge (when the cartridge is of the matrix type)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 1000

When the first cartridge is of the matrix type (bit 0 (MT1) of parameter No. 13240 is set to 1), set the number of rows in the pot in this parameter. The setting must satisfy the following condition, however: The sum total of the value obtained by (setting of parameter No. 13241) × (setting of parameter No. 13242) and the number of pots of other cartridges should not exceed 64 (1000 at maximum). If this condition is not satisfied or this parameter is set to 0, the first cartridge is invalid.

13242	Number of columns of the first cartridge (when the cartridge is of the matrix type)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 1000

When the first cartridge is of the matrix type (bit 0 (MT1) of parameter No. 13240 is set to 1), set the number of columns in the pot in this parameter. The setting must satisfy the following condition, however: The sum total of the value obtained by (setting of parameter No. 13241) × (setting of parameter No. 13242) and the number of pots of other cartridges should not exceed 64 (1000 at maximum). If this condition is not satisfied or this parameter is set to 0, the first cartridge is invalid.

13243	Number of rows of the second cartridge (when the cartridge is of the matrix type)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 1000

When the second cartridge is of the matrix type (bit 1 (MT2) of parameter No. 13240 is set to 1), set the number of rows in the pot in this parameter. The setting must satisfy the following condition, however: The sum total of the value obtained by (setting of parameter No. 13243) × (setting of parameter No. 13244) and the number of pots of other cartridges should not exceed 64 (1000 at maximum). If this condition is not satisfied or this parameter is set to 0, the second cartridge is invalid.

13244	Number of columns of the second cartridge (when the cartridge is of the matrix type)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 1000

When the second cartridge is of the matrix type (bit 1 (MT2) of parameter No. 13240 is set to 1), set the number of columns in the pot in this parameter. The setting must satisfy the following condition, however: The sum total of the value obtained by (setting of parameter No. 13243) × (setting of parameter No. 13244) and the number of pots of other cartridges should not exceed 64 (1000 at maximum). If this condition is not satisfied or this parameter is set to 0, the second cartridge is invalid.

13245	Number of rows of the third cartridge (when the cartridge is of the matrix type)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 1000

When the third cartridge is of the matrix type (bit 2 (MT3) of parameter No. 13240 is set to 1), set the number of rows in the pot in this parameter. The setting must satisfy the following condition, however: The sum total of the value obtained by (setting of parameter No. 13245) × (setting of parameter No. 13246) and the number of pots of other cartridges should not exceed 64 (1000 at maximum). If this condition is not satisfied or this parameter is set to 0, the third cartridge is invalid.

13246	Number of columns of the third cartridge (when the cartridge is of the matrix type)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 1000

When the third cartridge is of the matrix type (bit 2 (MT3) of parameter No. 13240 is set to 1), set the number of columns in the pot in this parameter. The setting must satisfy the following condition, however: The sum total of the value obtained by (setting of parameter No. 13245) × (setting of parameter No. 13246) and the number of pots of other cartridges should not exceed 64 (1000 at maximum). If this condition is not satisfied or this parameter is set to 0, the third cartridge is invalid.

13247	Number of rows of the fourth cartridge (when the cartridge is of the matrix type)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 1000

When the fourth cartridge is of the matrix type (bit 3 (MT4) of parameter No. 13240 is set to 1), set the number of rows in the pot in this parameter. The setting must satisfy the following condition, however: The sum total of the value obtained by (setting of parameter No. 13247) × (setting of parameter No. 13248) and the number of pots of other cartridges should not exceed 64 (1000 at maximum). If this condition is not satisfied or this parameter is set to 0, the fourth cartridge is invalid.

13248	Number of columns of the fourth cartridge (when the cartridge is of the matrix type)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 1000

When the fourth cartridge is of the matrix type (bit 3 (MT4) of parameter No. 13240 is set to 1), set the number of columns in the pot in this parameter. The setting must satisfy the following condition, however: The sum total of the value obtained by (setting of parameter No. 13247) × (setting of parameter No. 13248) and the number of pots of other cartridges should not exceed 64 (1000 at maximum). If this condition is not satisfied or this parameter is set to 0, the fourth cartridge is invalid.

Alarm and message

Number	Message	Description
PS5312	ILLEGAL COMMAND IN G10 L75/76/77	In commands from G10 L75/L76/L77 to G11, there is a format error, or a value beyond the specifiable range is specified. Correct the program.
PS5360	TOOL INTERFERENCE CHECK ERROR	This alarm is issued when interference with another tool is caused by a data modification based on G10 data input or file inputting or when an attempt is made to modify the tool figure data of a tool registered in the cartridge.
PS5361	ILLEGAL MAGAZINE DATA	Tools stored in the cartridge are interfering with each other. Reregister the tools in the cartridge, or modify the tool management data or tool figure data. If this alarm is issued, no tool interference check is made when tools are registered in the cartridge management table. Moreover, empty pot search operation does not operate normally. If this alarm is issued, the power must be turned off before operation is continued.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	TOOL MANAGEMENT FUNCTION FOR OVERSIZE TOOLS

11.4 TOOL COMPENSATION

11.4.1 Cutter Compensation and Tool Nose Radius Compensation

Overview

- Cutter compensation

Use of cutter compensation can offset a programmed tool path by the tool radius set in the CNC when machining is performed.
When the radius of the tool to be used for machining is measured and set as the offset value in the CNC, the tool moves along the offset path to cut a programmed profile. Therefore, even when the tool diameter changes, you must only change the offset value and need not modify the program.

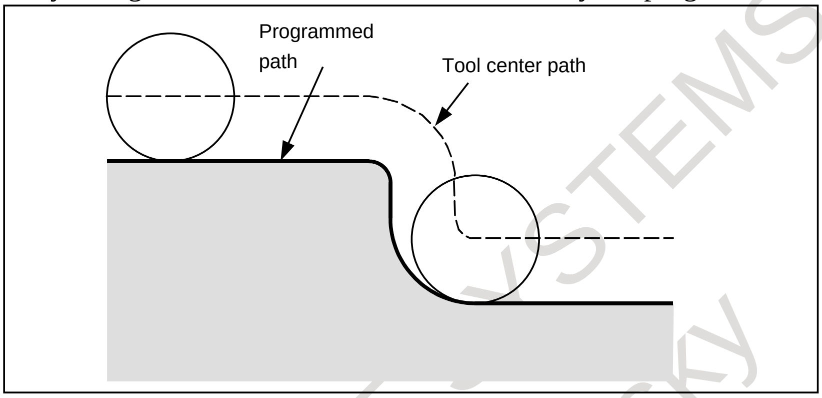


Fig.11.4.1 (a)

To obtain the actual offset tool path, the CNC internally calculates intersections of a straight line and a straight line, an arc and an arc, and a straight line and an arc automatically. The programmer only has to program a machining profile, therefore, programming can be done very easily.

- Tool nose radius compensation

Use of tool-nose radius compensation can offset a programmed tool path by the tool nose radius set in the CNC when machining is performed. When a machining profile is programmed using this function, and the radius of the tool nose to be used for actual machining is measured and set as the offset value in the CNC, the tool moves along the offset path to cut the programmed profile. Therefore, even when the tool nose radius changes, you must only change the offset value and need not modify the program.

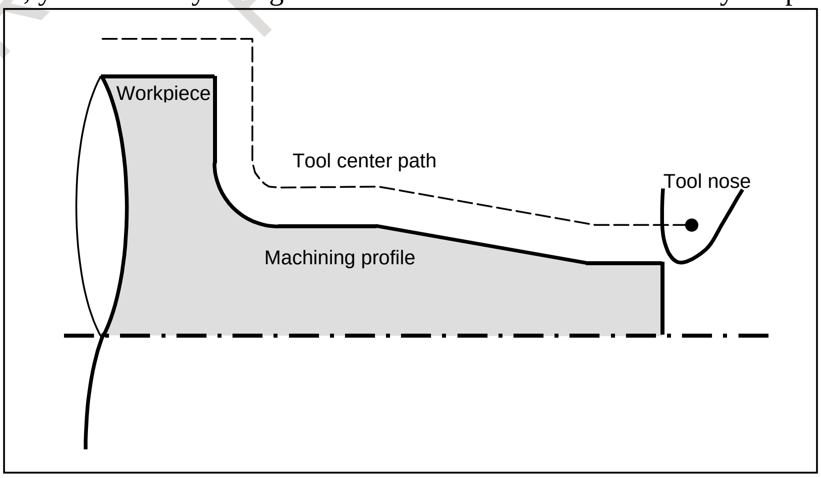


Fig.11.4.1 (b)

To obtain the actual offset tool path, the CNC internally calculates intersections of a straight line and a straight line, an arc and an arc, and a straight line and an arc automatically. The programmer only has to program a machining profile, therefore the programming can be done very easily.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5000								SBK

[Input type] Setting input
[Data type] Bit path

#0 SBK With a block created internally for tool radius - tool nose radius compensation:
0: A single block stop is not performed.
1: A single block stop is performed.
This parameter is used to check a program including cutter compensation/tool nose radius compensation.

	#7	#6	#5	#4	#3	#2	#1	#0
5001								
				EVR				

[Input type] Parameter input
[Data type] Bit path

#4 EVR When a tool compensation value is changed in tool radius - tool nose radius compensation mode:
0: Enables the change, starting from that block where the next D or H code is specified.
1: Enables the change, starting from that block where buffering is next performed.

	#7	#6	#5	#4	#3	#2	#1	#0
5002	WNP						LGN	

[Input type] Parameter input
[Data type] Bit path

#1 LGN Geometry offset number of tool offset
0: Is the same as wear offset number
1: Specifies the geometry offset number by the tool selection number

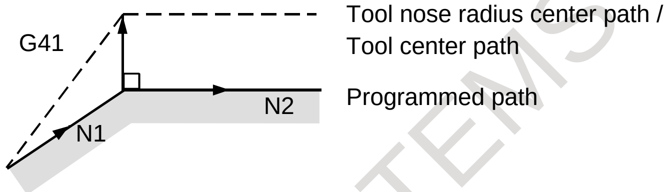
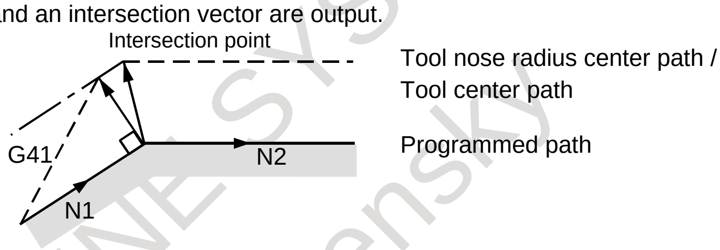
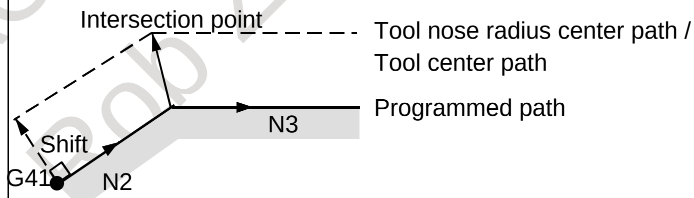
> **NOTE**
> This parameter is valid when tool geometry/wear compensation is specified.

#7 WNP Imaginary tool tip number used for tool nose radius compensation, when the tool geometry/wear compensation function is equipped, is the number specified by:
0: Geometry offset number
1: Wear offset number

	#7	#6	#5	#4	#3	#2	#1	#0
5003							**SUV**	**SUP**

[Input type] Parameter input
[Data type] Bit path

#0 SUP
#1 SUV These bits are used to specify the type of startup/cancellation of tool radius - tool nose radius compensation.

<table>
<tr><th>SUV</th><th>SUP</th><th>Type</th><th>Operation</th></tr>
<tr><td>0</td><td>0</td><td>Type A</td><td>A compensation vector perpendicular to the block next to the startup block or the block preceding the cancellation block is output.

</td></tr>
<tr><td>0</td><td>1</td><td>Type B</td><td>A compensation vector perpendicular to the startup block or cancellation block and an intersection vector are output.

</td></tr>
<tr><td>1</td><td>0
1</td><td>Type C</td><td>When the startup block or cancellation block specifies no movement operation, the tool is shifted by the cutter compensation amount in a direction perpendicular to the block next to the startup or the block before cancellation block.


When the block specifies movement operation, the type is set according to the SUP setting; if SUP is 0, type A is set, and if SUP is 1, type B is set.</td></tr>
</table>

> **NOTE**
> When SUV,SUP = 0,1 (type B), an operation equivalent to that of FS16*i*-T is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
5004								
						ODI		

[Input type] Parameter input
[Data type] Bit path

#2 ODI The setting of a tool radius - tool nose radius compensation value is corrected as:
0: Radius value
1: Diameter value

	#7	#6	#5	#4	#3	#2	#1	#0
5008				MCR	CNV		CNC	

[Input type] Parameter input
[Data type] Bit path

#1 CNC
#3 CNV These bits are used to select an interference check method in the tool radius - tool nose radius compensation mode.

CNV	CNC	Operation
0	0	Interference check is enabled. The direction and the angle of an arc are checked.
0	1	Interference check is enabled. Only the angle of an arc is checked.
1	-	Interference check is disabled.

For the operation taken when the interference check shows the occurrence of an reference (overcutting) , see the description of bit 5 (CAV) of parameter No. 19607.

NOTE
Checking of only the direction cannot be set.

#4 MCR If G41/G42 (tool radius - tool nose radius compensation) is specified in the MDI mode, an alarm is:
0: Not raised.
1: Raised. (Alarm PS5257 “G41/G42 NOT ALLOWED IN MDI MODE”)

	#7	#6	#5	#4	#3	#2	#1	#0
5009								
			TIP					

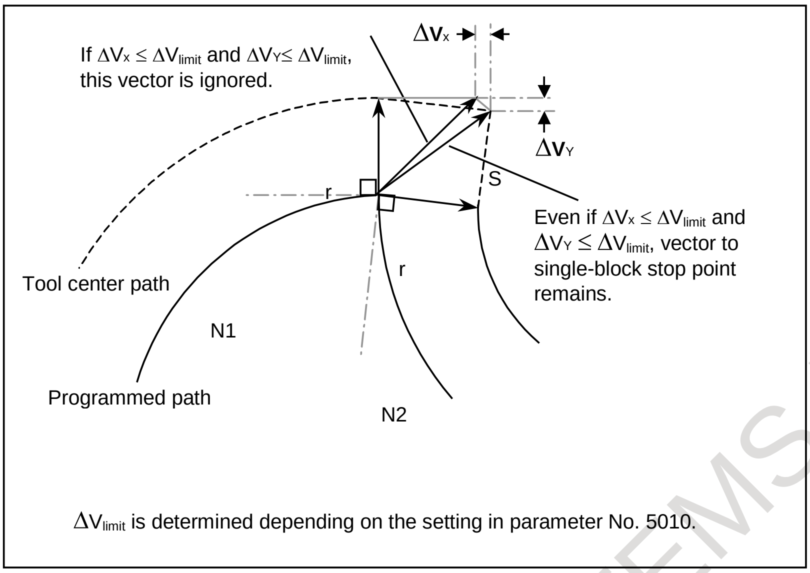
[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#5 TIP In tool radius - tool nose radius compensation, the virtual tool tip direction is:
0: Not used.
1: Used.

5010	Limit for ignoring the small movement resulting from tool radius - tool nose radius compensation

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
When the tool moves around a corner in cutter compensation or tool nose radius compensation mode, the limit for ignoring the small travel amount resulting from compensation is set. This limit eliminates the interruption of buffering caused by the small travel amount generated at the corner and any change in feedrate due to the interruption.



5024	Number of tool compensation values

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 999

Set the maximum allowable number of tool compensation values used for each path.
Ensure that the total number of values set in parameter No. 5024 for the individual paths is within the number of compensation values usable in the entire system. The number of compensation values usable in the entire system depends on the option configuration.
If the total number of values set in parameter No. 5024 for the individual paths exceeds the number of compensation values usable in the entire system, or 0 is set in parameter No. 5024 for all paths, the number of compensation values usable for each path is a value obtained by dividing the number of compensation values usable in the entire system by the number of paths.
Tool compensation values as many as the number of compensation values used for each path are displayed on the screen. If tool compensation numbers more than the number of compensation values usable for each path are specified, an alarm PS0115 “VARIABLE NO. OUT OF RANGE” is issued.
For example, 64 tool compensation sets are used, 20 sets may be allocated to path 1, 30 sets to path 2, and 14 sets to path 3. All of 64 sets need not be used.

	#7	#6	#5	#4	#3	#2	#1	#0
5042							OFC	OFA

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 OFA

#1 OFC These bits are used to specify the increment system and valid data range of a tool offset value.

For metric input

OFC	OFA	Unit	Valid data range
0	1	0.01mm	±9999.99mm
0	0	0.001mm	±9999.999mm
1	0	0.0001mm	±9999.9999mm

For inch input

OFC	OFA	Unit	Valid data range
0	1	0.001inch	±999.999inch
0	0	0.0001inch	±999.9999inch
1	0	0.00001inch	±999.99999inch

	#7	#6	#5	#4	#3	#2	#1	#0
19607	NAG	NAA	CAV			CCC		

[Input type] Parameter input
[Data type] Bit path

#2 CCC In the cutter compensation/tool nose radius compensation mode, the outer corner connection method is based on:
0: Linear connection type.
1: Circular connection type.

#5 CAV When an interference check for cutter compensation/tool nose radius compensation finds that interference (overcutting) occurred:
0: Machining stops with the alarm PS0041," INTERFERENCE IN CUTTER COMPENSATION".
(Interference check alarm function)
1: Machining is continued by changing the tool path to prevent interference (overcutting) from occurring. (Interference check avoidance function)
For the interference check method, see the descriptions of bit 1 (CNC) of parameter No. 5008 and bit 3 (CNV) of parameter No. 5008.

#6 NAA When the interference check avoidance function for cutter compensation/tool nose radius compensation considers that an avoidance operation is dangerous or that a further interference to the interference avoidance vector occurs:
0: An alarm is issued.
When an avoidance operation is considered to be dangerous, the alarm PS5447," DANGEROUS AVOIDANCE AT G41/G42" is issued.
When a further interference to the interference avoidance vector is considered to occur, the alarm PS5448," INTERFERENCE TO AVD. AT G41/G42" is issued.
1: No alarm is issued, and the avoidance operation is continued.

> ⚠ **CAUTION**
> When this parameter is set to 1, the path may be shifted largely. Therefore, set this parameter to 0 unless special reasons are present.

#7 NAG If the gap vector length is 0 when the interference check avoidance function for cutter compensation/tool nose radius compensation is used:
0: Avoidance operation is performed.

1: Avoidance operation is not performed.

19625	Number of blocks to be read in the cutter compensation/tool nose radius compensation mode

[Input type] Setting input
[Data type] Byte path
[Valid data range] 3 to 8

This parameter sets the number of blocks to be read in the cutter compensation/tool nose radius compensation mode. When a value not greater than 3 is set, the specification of 3 is assumed. When a value not less than 8 is set, the specification of 8 is assumed. As a greater number of blocks are read, an overcutting (interference) forecast can be made with a command farther ahead. However, the number of blocks read and analyzed increases, so that a longer block processing time becomes necessary.

Even if the setting of this parameter is modified in the MDI mode by stopping in the cutter compensation/tool nose radius compensation mode, the setting does not become valid immediately. Before the new setting of this parameter can become valid, the cutter compensation/tool noise radius compensation mode must be canceled, then the mode must be entered again.

Alarm and message

Number	Message	Description
PS0033	NO INTERSECTION AT 41/42	The intersection cannot be obtained by the intersection calculation in cutter or tool-nose radius compensation. Modify the program.
PS0034	ONLY G00/G01 ALLOWED IN STUP/EXT BLK	In cutter or tool-nose radius compensation, a startup or cancellation is performed in the following mode. Modify the program. G02/G03 : Circular interpolation or helical interpolation G35/G36 : Circular threading
PS0035	CAN NOT COMMANDED G31	- G31 cannot be specified. This alarm is generated when a G code (such as for cutter or tool-nose radius compensation) of group 07 is not canceled. - A torque limit skip was not specified in a torque limit skip command (G31P98 or P99). Specify the torque limit skip in the PMC window or the like. Or, specify the torque limit override by address Q.
PS0037	CAN NOT CHANGE PLANE IN G41/G42	The compensation plane G17/G18/G19 was changed during cutter or tool-nose radius compensation. Modify the program.
PS0041	INTERFERENCE IN G41/42	In cutter or tool-nose radius compensation, excessive cutting may occur. Modify the program.
PS0042	G45/G48 NOT ALLOWED IN CRC	Tool offset (G45 to G48) is commanded in tool compensation. Modify the program.
PS5445	CAN NOT COMMAND MOTION IN G39	Corner circular interpolation (G39) of cutter compensation or tool nose radius compensation is not specified alone but is specified with a move command.
PS5446	NO AVOIDANCE AT G41/G42	Because there is no interference evade vector, the interference check evade function of cutter compensation or tool nose radius compensation cannot evade interference.
PS5447	DANGEROUS AVOIDANCE AT G41/G42	The interference check evade function of cutter compensation or tool nose radius compensation determines that an evade operation will lead to danger.

Number	Message	Description
PS5448	INTERFERENCE TO AVD. AT G41/G42	In the interference check evade function of cutter compensation or tool nose radius compensation, a further interference occurs for an already created interference evade vector.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (For lathe system) (B-64694EN-1)	Overview of Tool Nose Radius Compensation (G40-G42) Overview of Cutter Compensation (G40-G42) Details of Cutter or Tool Nose Radius Compensation
OPERATOR'S MANUAL (For machining center system) (B-64694EN-2)	Overview of Tool Nose Radius Compensation (G40-G42) Overview of Cutter Compensation (G40-G42) Details of Cutter or Tool Nose Radius Compensation

11.4.2 Tool Length Compensation

M

Overview

When the difference between the tool length assumed at the time of programming and the tool length of the tool actually used for machining is set in offset memory, the difference in tool length can be corrected without modifying the program.
G43 and G44 specify the offset direction, and a number following the tool length compensation specification address (H code) specifies the tool length compensation amount set in the offset memory.

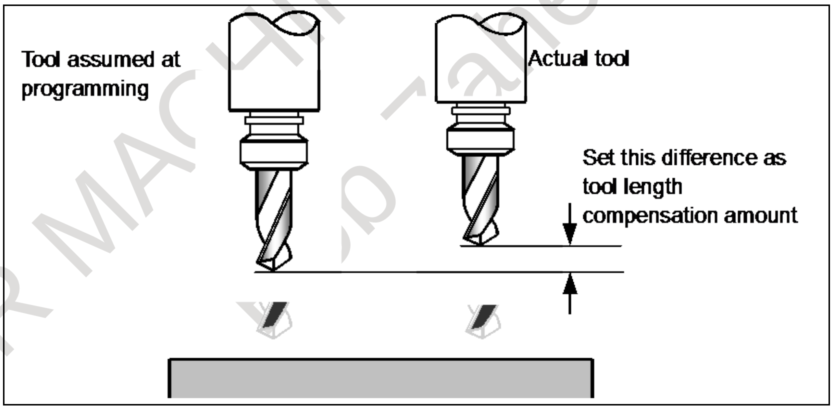


Fig.11.4.2 (a)

One of the following three methods is available, depending on the type of axis that can be subject to tool length compensation:

- Tool length compensation A
  Compensates the value of the tool length on the Z axis.
- Tool length compensation B
  Compensates the value of the tool length on vertical direction of plane selection .
- Tool length compensation C
  Compensates the value of the tool length on a specified axis.

T

Extended tool selection function is necessary to use this function with lathe system. Moreover, the tool change method needs to be ATC type (bit 3(TCT) of parameter No.5040) = 1).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3104								
		DAL		DRL				

[Input type] Parameter input
[Data type] Bit path

#4 DRL Relative position
0: The actual position displayed takes into account tool length offset.
1: The programmed position displayed does not take into account tool length offset.

#6 DAL Absolute position
0: The actual position displayed takes into account tool length offset.
1: The programmed position displayed does not take into account tool length offset.

> **NOTE**
> In lathe systems, whether to exclude a tool offset when displaying the absolute position is determined by the setting of bit 1 (DAP) of parameter No. 3129.

	#7	#6	#5	#4	#3	#2	#1	#0
5001		EVO						
		EVO			TAL		TLB	TLC

[Input type] Parameter input
[Data type] Bit path

#0 TLC
#1 TLB These bits are used to select a tool length compensation type.

Type	TLB	TLC
Tool length compensation A	0	0
Tool length compensation B	1	0
Tool length compensation C	-	1

The axis to which cutter compensation is applied varies from type to type as described below.
Tool length compensation A : Z-axis at all times
Tool length compensation B : Axis perpendicular to a specified plane (G17/G18/G19)
Tool length compensation C : Axis specified in a block that specifies G43/G44

#3 TAL Tool length compensation C
0: Generates an alarm when two or more axes are offset
1: Not generate an alarm even if two or more axes are offset

#6 EVO If a tool compensation value modification is made for tool length compensation A or tool length compensation B in the offset mode (G43 or G44):
0: The new value becomes valid in a block where G43, G44, or an H code is specified next.
1: The new value becomes valid in a block where buffering is performed next.

	#7	#6	#5	#4	#3	#2	#1	#0
5003		LVK						

[Input type] Parameter input
[Data type] Bit path

#6 LVK By reset, Tool length compensation vector is
0: Canceled
1: Not canceled.

	#7	#6	#5	#4	#3	#2	#1	#0
5007	3OF	3OC						

[Input type] Parameter input
[Data type] Bit path

#6 3OC If tool length compensation is not cancelled before 3-dimensional coordinate conversion is specified, an alarm is:
0: Not raised.
1: Raised. (alarm PS0049 ,” ILLEGAL COMMAND (G68,G69)”)

#7 3OF If 3-dimensional coordinate conversion is not nested with a command for tool length compensation, or if 3-dimensional coordinate conversion is specified during tool length compensation and another command for tool length compensation is specified:
0: No alarm is issued.
1: The alarm PS0049 ,” ILLEGAL COMMAND (G68,G69)” is issued.

Example 1) An alarm is raised in the following cases:

```
1) G68 X_ Y_ Z_ I_ J_ K_ R_ ;
      :
   G43 H1 ;
      :
   G69 ;                              ← PS0049 raised
      :
   G49 ;
2) G43 H1 ;
      :
   G68 X_ Y_ Z_ I_ J_ K_ R_ ;
      :
   G49(G43H2,H2 etc.) ;               ← PS0049 raised
      :
   G69 ;
```

Example 2) No alarm is raised in the following cases:

```
3) G68 X_ Y_ Z_ I_ J_ K_ R_ ;
     :
   G43 H1 ;
     :
   G49 ;
     :
   G69 ;
4) G43 H1 ;
     :
   G68 X_ Y_ Z_ I_ J_ K_ R_ ;
     :
   G69 ;
     :
   G49 ;
```

NOTE

A command to cancel tool length compensation (G28, etc.) will not cause an alarm to be raised. If a command like this is specified in the G68 mode, program as indicated in 3) above.

```
G43 H1 ;
  :
G68 X_ Y_ Z_ I_ J_ K_ R_ ;
  :
G28 X_ Y_ Z_ ;            ← Offset is cancelled.
  :                         No alarm is raised.
G69 ;
```

Alarm and message

Number	Message	Description
PS0027	NO AXES COMMANDED IN G43/G44	No axis is specified in G43 and G44 blocks for the tool length offset type C. Offset is not canceled but another axis is offset for the tool length offset type C. Multiple axes were specified for the same block when the tool length compensation type is C.
PS0049	ILLEGAL COMMAND(G68,G69)	When 3-dimensional coordinate conversion (G68 or G69) was specified, the tool compensation was not canceled. Or, programs of 3-dimensional coordinate conversion (G68, G69) and tool compensation (G43, G44 or G49) were not nested. Or, the 3-dimensional coordinate conversion was specified during the tool length compensation and another tool length compensation was specified.
PS0336	TOOL COMPENSATION COMMANDED MORE TWO AXES	For a tool length compensation C, an attempt was made to command the offset to other axes without canceling the offset. Or, for a tool length compensation C, multiple axes are specified in G43 or G44 block.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Tool length compensation

11.4.3 Tool Length Compensation Shift Types

M

Overview

A tool length offset operation can be performed by shifting the program coordinate system. The coordinate system containing the axis subject to tool length compensation is shifted by the tool length compensation value. A tool length compensation shift type can be selected with bit 6 (TOS) of parameter No. 5006 or bit 2 (TOP) of parameter No. 11400. If no move command is specified together with the G43, G44, or G49 command, the tool will not move along the axis. If a move command is specified together with the G43, G44, or G49 command, the coordinate system will be shifted first, then the tool will move along the axis.
One of the following three methods is available, depending on the type of axis that can be subject to tool length compensation:

- Tool length compensation A
  Compensates the value of the tool length on the Z axis.
- Tool length compensation B
  Compensates the value of the tool length on one of the X, Y, and Z axis.
- Tool length compensation C
  Compensates the value of the tool length on a specified axis.

Explanation

- Offset direction

If the tool length compensation value specified with an H code (and stored in offset memory) is G43, the coordinate system is shifted to the + side; if G44, to the - side. If the sign of the tool length compensation value is -, the coordinate system is shifted to the - side if G43 and to the + side if G44. G43 and G44 are modal G codes; they remain valid until another G code in the same group is used.

- Specifying a tool length compensation value

The tool length compensation value corresponding to the number (offset number) specified with an H code (and stored in offset memory) is used. The tool length compensation corresponding to the offset number 0 always means 0. It is not possible to set a tool length compensation value corresponding to H0.

- Compensation axis

Specify one of tool length compensation types A, B, and C, using bits 0 (TLC) and 1 (TLB) of parameter No. 5001.

- Specifying offset on two or more axes

Tool length compensation B enables offset on two or more axes by specifying offset axes in multiple blocks.

To perform offset on X and Y axes

G19 G43 H_;	Performs offset on the X axis.
G18 G43 H_;	Performs offset on the Y axis.

Tool length compensation C suppresses the generation of an alarm even if offset is performed on two or more axes at the same time, by setting bit 3 (TAL) of parameter No. 5001 to 1.

- Tool length compensation cancel

To cancel offset, specify either G49 or H0. Canceling offset causes the shifting of the coordinate system to be undone. If no move command is specified at this time, the tool will not move along the axis.

- Staring and canceling tool length compensation

Once a command for starting or canceling tool length compensation (*2) is issued in a mode such as cutter compensation (*1), no look-ahead block interpretation is carried out until the tool length compensation start or cancel request block is finished. So, the following operations occur.

- The spindle decelerates to a stop in the start or cancel block.
- The compensation vector for tool compensation becomes perpendicular to the block just before the start or cancel block, resulting in an overcut or insufficient cut, because no look-ahead interpretation is carried out.
- No command specified after the start or cancel block is executed until the block is finished.

*1 No look-ahead block interpretation is carried out for the following commands:
  - G codes, except G40, in group 07
    (for such modes as cutter compensation (G41/G42))

*2 The following commands are included.
  - Tool length compensation (G43/G44)

- Operation when a tool compensation value is changed in the tool length compensation mode

Bit 1 (MOF) of parameter No. 5000 can be used to specify what operation to perform when a tool length compensation value is changed (*3) in the cutter compensation mode (*1) and tool length compensation mode (*2).

- If bit 1 (MOF) of parameter No. 5000 = 0:
  The axis is shifted by the amount of change in the tool length compensation value.
- If bit 1 (MOF) of parameter No. 5000 = 1:
  Until an absolute command is issued to the compensated axis since the change of the tool length compensation value, the axis is shifted by the amount of change in the tool length compensation value.

*1 The following commands are included.
  - G codes, except G40, in group 07
    Cutter compensation (G41/G42)

*2 The following commands are included.
  - Tool length compensation (G43/G44)

*3 "Changing a tool length compensation value" pertains to:
  - Issuing the H code (or, with the extended tool select function in the lathe system, D code) in a program
  - Changing a compensation value on the offset screen, by the G10 command, system variables, or the window function if bit 6 (EVO) of parameter No. 5001 = 1.

Caution

⚠ **CAUTION**

1 Specifying tool length compensation (a shift type) first and then executing an incremental command causes the tool length compensation value to be reflected in the coordinates only, not in the travel distance of the machine; executing an absolute command causes the tool length compensation value to be reflected in both the movement of the machine and the coordinates.
2 If a programmable mirror image is effective, the tool length offset is applied in the specified direction.
3 No scaling magnification is applied to the tool length offset value.
4 No coordinate system rotation is applied to the tool length offset value. Tool length offset is effective in the direction in which the offset is applied.
5 The tool length offset operation is independent of the cutter compensation offset operation.
6 3-dimensional coordinate conversion is applied to tool length offset. If tool length offset is made effective to multiple axes, the tool length offset must be canceled for one axis at a time.
7 With the WINDOW command, changing bit 6 (TOS) of parameter No. 5006 or bit 2 (TOP) of parameter No. 11400 during automatic operation does not cause the tool length offset type to be changed.
8 If offset has been performed on two or more axes with tool length compensation B, a G49 command causes the offset to be canceled on all axes; H0 causes the offset to be canceled only on the axis vertical to the specified plane.
9 If the tool length compensation value is changed by changing the offset number, this simply means that the value is replaced by a new tool length compensation value; it does not mean that a new tool length compensation value is added to the old tool length compensation.
10 If using cutter compensation, set bit 2 (OFH) of parameter No. 5001 to 0, specify tool length compensation with an H code, and specify cutter compensation with a D code.
11 If reference position return (G28 or G30) has been specified, tool length offset is canceled for the axis specified at the time of positioning on the reference point; however, tool length offset is not canceled for an un-specified axis. If reference position return has been specified in the same block as that containing tool length offset cancel (G49), tool length offset is canceled for both the specified and un-specified axes at the time of positioning on the mid-point.
12 With a machine coordinate system command (G53), tool length offset is canceled for the axis specified at the time of positioning on the specified point.
13 The tool length compensation vector canceled by specifying G53, G28, or G30 during tool length compensation is restored as described below:
For tool length compensation types A and B, if bit 6 (EVO) of parameter No. 5001 is 1, the vector is restored in the block buffered next; for all of tool length compensation types A, B, and C, it is restored in a block containing an H, G43, or G44 command if parameter is 0.
14 When a tool compensation shift type is used, no look-ahead interpretation is made if a command for starting or canceling compensation such as tool length compensation is issued in the tool radius/ tool-nose radius compensation mode. As a result, it is likely that an overcut or insufficient cut may occur. To avoid this problem, issue the command before the tool radius/ tool-nose radius compensation mode is entered or in a place where machining is not affected.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5000							MOF	

[Input type] Setting input
[Data type] Bit path

#1 MOF When the tool length compensation shift type (bit 6 (TOS) is set to 1 of parameter No. 5006 or bit 2 (TOP) of parameter No. 11400 is set to 1) is used, if the tool length compensation amount is changed(NOTE 3) in the tool length compensation mode (NOTE 1) when look-ahead blocks are present(NOTE 2):

0: Compensation is performed for the change in compensation amount as the movement type.

1: Compensation is not performed for the change until a tool length compensation command (offset number) and an absolute programming for the compensation axis are specified.

> **NOTE**
> 1 The tool length compensation mode refers to the following state:
> - Tool length offset (G43/G44)
>
> 2 "When look-ahead blocks are present" means as follows:
> - The modal G code of the G codes (such as tool radius - tool nose radius compensation) of group 07 is other than G40.
>
> One look-ahead block during automatic operation and multiple look-ahead blocks in the AI contour control mode are not included in the state "when look-ahead blocks are present".
>
> 3 Changes in tool length compensation amount are as follows:
> - When the tool length compensation number is changed by H code (or D code for the extended tool selection function for lathe systems)
> - When G43 or G44 is specified to change the direction of tool length compensation
> - When the tool length compensation amount is changed using the offset screen, G10 command, system variable, PMC window, and so forth during automatic operation if bit 1 (EVO) of parameter No. 5001 is set to 1.
> - When the tool length compensation vector is restored after being temporarily canceled by G53, G28, or G30 during tool length compensation.

	#7	#6	#5	#4	#3	#2	#1	#0
5001								
					TAL		TLB	TLC

[Input type] Parameter input
[Data type] Bit path

#0 TLC
#1 TLB These bits are used to select a tool length compensation type.

Type	TLB	TLC
Tool length compensation A	0	0
Tool length compensation B	1	0

Type	TLB	TLC
Tool length compensation C	-	1

The axis to which cutter compensation is applied varies from type to type as described below.
Tool length compensation A : Z-axis at all times
Tool length compensation B : Axis perpendicular to a specified plane (G17/G18/G19)
Tool length compensation C : Axis specified in a block that specifies G43/G44

#3 TAL Tool length compensation C
0: Generates an alarm when two or more axes are offset
1: Not generate an alarm even if two or more axes are offset

	#7	#6	#5	#4	#3	#2	#1	#0
5006		TOS						

[Input type] Parameter input
[Data type] Bit

#6 TOS Set a tool length compensation or tool offset operation.
0: Tool length compensation or tool offset operation is performed by an axis movement.
1: Tool length compensation or tool offset operation is performed by shifting the coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
11400						TOP		

[Input type] Parameter input
[Data type] Bit path

#2 TOP Set a tool length compensation or tool offset operation.
0: Tool length compensation or tool offset operation is performed by an axis movement.
1: Tool length compensation or tool offset operation is performed by shifting the coordinate system.

NOTE
This parameter is an-individual path parameter having the same function as bit 6 (TOS) of parameter No. 5006.
To use different compensation types for individual paths, set the parameter TOS with 0 and specify a compensation type for each path separately, using the parameter TOP. If the parameter TOS is 1, the parameter TOP is assumed to be 1 even if it is 0.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (For Machining Center System)(B-64694EN-2)	Tool length compensation shift types

11.4.4 Second Geometry Tool Offset

T

Overview

To compensate for the difference in tool mounting position and in selected position, this function adds 32 sets of second geometry tool offset for each of the X, Z, and Y axes for all paths (up to 2000 sets for all paths and up to 999 sets for a single path).
In contrast to this offset, the ordinary tool geometry offset is called the first geometry tool offset.
It is possible to apply a tool offset value (tool wear offset + tool geometry offset) in the reverse direction, using an appropriate signal.
This function may be used if the compensation value differs even with a single tool for a mechanical reason depending on the mounting position (inside/outside) or the selected position (right/left).

NOTE
1 To use the second geometry tool offset, the geometry and wear compensation are required.
2 To use the second geometry tool offset for the Y axis, the Y axis offset is required.

Example

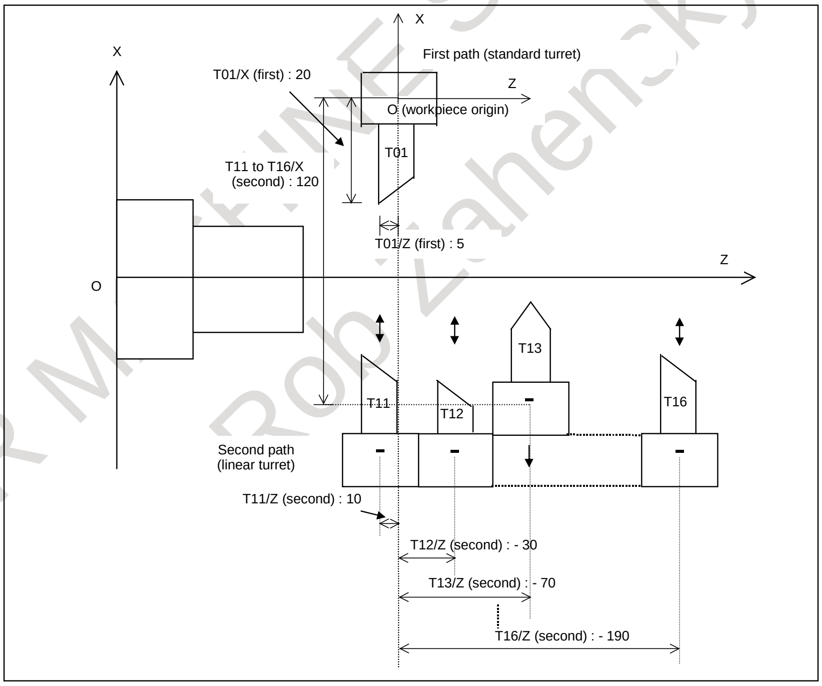


Fig. 11.4.4

In the machine configuration shown in the Fig. 11.4.4, for the offset data for the tool mounted to the first path (standard turret), set the offset data of the tool itself as the first geometry tool offset data. (The second geometry tool offset data is 0.) For the offset data for the tool mounted to the second path (linear turret), set the offset data of the tool itself as the first geometry tool offset data and the offset data from the workpiece origin at the mounting position as the second geometry tool offset data. Usually, the offset data of the tool itself is measured separately from the offset data at the mounting position, these data can be set separately by using the second geometry tool offset.

Table 11.4.4

First geometry tool offset			Second geometry tool offset		
No.	**X axis**	**Z axis**	**No.**	**X axis**	**Z axis**
01	20.000	5.000	01	0.000	0.000
:	:	:	:	:	:
10	25.000	8.000	10	0.000	0.000
11	-20.000	5.000	11	120.000	10.000
12	-10.000	3.000	12	120.000	-30.000
13	-15.000	0.000	13	120.000	-70.000
:	:	:	:	:	:
16	-18.000	7.000	16	120.000	-190.000

- Offset data

The data for the second geometry tool offset can be set for each path. The number of items of the data can be set with parameter No. 5024. The data remains stored even after the power is turned off.
If the data is to be made common to paths, use the memory common to paths.

Signal

Second geometry tool offset signal G2SLC<Gn090.7>

[Classification] Input signal

[Function] This signal selects whether to use the second geometry tool offset.

[Operation] When a tool number is specified with a T code, only the first geometry tool offset is used as the tool geometry offset value if this signal is 0; if this signal is 1, the value obtained by adding the second geometry tool offset to the first geometry tool offset is used as the tool geometry offset value.

> **NOTE**
> To switch this signal, use an M code for suppressing buffering. If the signal is switched without suppressing buffering during automatic operation, it is likely that no offset may work properly.

Second geometry tool offset axis selection signals G2X,G2Z,G2Y<Gn090.4, Gn090.5, Gn090.6>

[Classification] Input signal

[Function] These signals select the axis for which the second geometry tool offset is added to the first geometry tool offset.

[Operation] If these signals are set to 0, the second geometry tool offset is not added to the controlled axis corresponding to the axis bit.
If these signals are set to 1, the second geometry tool offset is added to the controlled axis corresponding to the axis bit.
These signals are valid when the second geometry tool offset signal G2SLC is set to 1.

> **NOTE**
> G2Y requires the Y-axis offset.

Tool offset direction signals G2RVX,G2RVZ,G2RVY<Gn090.0, Gn090.1, Gn090.2>

[Classification] Input signal

[Function] When a tool offset is used, these signals change the direction of the compensation amount.
The direction of the tool offset for compensation is determined by tool movement as follows:
0: The same direction as the direction indicated by the sign

1: The opposite direction to the direction indicated by the sign

> **NOTE**
> 1 G2RVY requires the Y-axis offset.
> 2 This signal cannot shift a coordinate.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn090	G2SLC	G2Y	G2Z	G2X		G2RVY	G2RVZ	G2RVX

Parameter

3032	Allowable number of digits for the T code

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 8
Set the allowable numbers of digits for the T code.
When 0 is set, the allowable number of digits is assumed to be 8.

	#7	#6	#5	#4	#3	#2	#1	#0
3290			GO2					

[Input type] Parameter input
[Data type] Bit path

#5 GO2 Setting the second geometric tool offset value by MDI key input is:
0: Disabled.
1: Not disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5002		LWM		LGT			LGN	

[Input type] Parameter input
[Data type] Bit path

#1 LGN Geometry offset number of tool offset
0: Is the same as wear offset number
1: Specifies the geometry offset number by the tool selection number

> **NOTE**
> This parameter is valid when tool geometry/wear compensation is specified.

#4 LGT Tool geometry compensation
0: Compensated by the shift of the coordinate system
1: Compensated by the tool movement

> **NOTE**
> This parameter is valid when tool geometry/wear compensation is specified.

#6 LWM Tool offset operation based on tool movement is performed:
0: In a block where a T code is specified.
1: Together with a command for movement along an axis.

5024	Number of tool compensation values

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 999

Set the maximum allowable number of tool compensation values used for each path.
Ensure that the total number of values set in parameter No. 5024 for the individual paths is within the number of compensation values usable in the entire system. The number of compensation values usable in the entire system depends on the option configuration.
If the total number of values set in parameter No. 5024 for the individual paths exceeds the number of compensation values usable in the entire system, or 0 is set in parameter No. 5024 for all paths, the number of compensation values usable for each path is a value obtained by dividing the number of compensation values usable in the entire system by the number of paths.
Tool compensation values as many as the number of compensation values used for each path are displayed on the screen. If tool compensation numbers more than the number of compensation values usable for each path are specified, alarm PS0115 “VARIABLE NO. OUT OF RANGE” is issued.
For example, 64 tool compensation sets are used, 20 sets may be allocated to path 1, 30 sets to path 2, and 14 sets to path 3. All of 64 sets need not be used.

5028	Number of digits of an offset number used with a T code command

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 4

Specify the number of digits of a T code portion that is used for a tool offset number (wear offset number when the tool geometry/wear compensation function is used).
When 0 is set, the number of digits is determined by the number of tool compensation values.
When the number of tool compensation values is 1 to 9: Lower 1 digit
When the number of tool compensation values is 10 to 99: Lower 2 digits
When the number of tool compensation values is 100 to 999: Lower 3 digits

[Example] When an offset number is specified using the lower 2 digits of a T code, set 2 in parameter No. 5028.
Txxxxxx yy
xxxxxx : Tool selection
yy : Tool offset number

In tool management function, set 4 in this parameter to set 4-digit number in parameter No.13265.

> **NOTE**
> A value longer than the setting of parameter No. 3032 (allowable number of digits of a T code) cannot be set.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (For Lathe System) (B-64694EN-1)	Second Geometry Tool Offset

11.5 TOOL LIFE MANAGEMENT

Tools are classified into several groups, and a tool life (use count or use duration) is specified for each group in advance. Each time a tool is used, its life is counted, and when the tool life expires, a new tool that is sequenced next within the same group is selected automatically. With this function, the tool life can be managed while machining is being performed continuously. Data for tool life management consists of tool group numbers, tool life values, tool numbers, and codes for specifying a tool offset value. These data items are registered in the CNC.

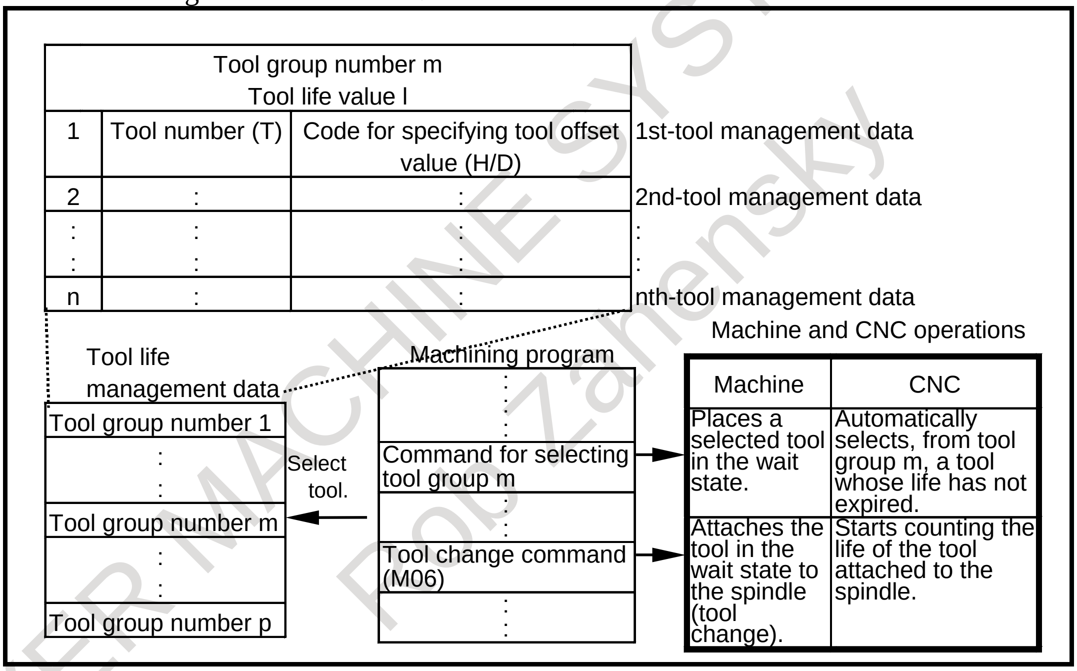


Fig. 11.5 (a) Tool selection from machining program

M

A group is selected by a T code, and tool life counting is started by the M06 command.

T

The T series has two tool change types (the turret type and the ATC type). The turret type uses only a T code to select a group, specify a tool offset value, and start tool life counting, whereas the ATC type, like the M series, uses a T code to select a group and the M06 command to start tool life counting. With the ATC type, only a D code is used for specifying a tool offset value. The tool change type is selected by bit 3 (TCT) of parameter No. 5040.

> ⚠ **CAUTION**
> This function cannot be used if bit 1 (LGN) of parameter No. 5002 is set to 1 to use the same number as the tool selection number to specify a geometric offset number.

- Life management B function

If the tool life management B function is enabled, the maximum tool life value can be extended, and the tool life expiration prior notice signal TLCHB <Fn064.3> can be output to post tool life expiration in advance when the remaining life (the life value - the life counter value) has reached the remaining life setting. The remaining life setting is registered as tool life management data in the CNC in advance.
The tool life management B function is enabled by setting bit 4 (LFB) of parameter No. 6805 to 1.

M

If the tool life management B function is enabled, the function for selecting a tool group by an arbitrary group number can be used.

T

The tool life management B function can be used. However, the function for selecting a tool group by an arbitrary group number can be used only if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1).

- Maximum number of tool life management groups and multi-path system

Up to 256 tool life management groups can be used in the entire CNC system.
For each path, set a maximum number of groups to be used in parameter No. 6813.
The maximum number of groups must be a multiple of the minimum number of groups (eight groups). If the maximum number of groups is 0, the tool life management function is disabled.

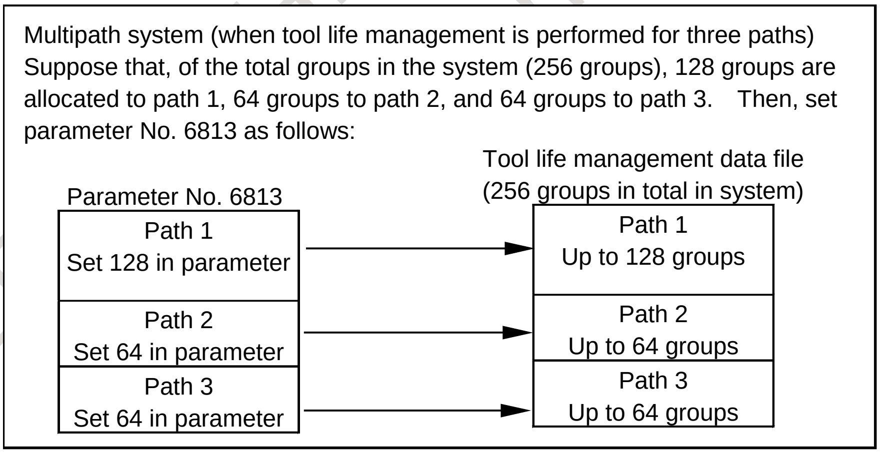


Fig. 11.5 (b) Group allocation in entire system

Signal

Tool change signal TLCH <Fn064.0>

[Classification] Output signal
[Function] Informs that the life of the last tool in a group has expired.
[Output cond.] The signal becomes 1 when:
- The life of the last tool in a group has expired, after the next tool of the group was selected each time the life of each tool in the group expired.

The signal becomes 0 when:
- There is no group whose life has expired.

When the signal is 1, it is reset to 0 by informing the CNC that tool change has been finished for all groups in which the life of all tools had expired, by inputting the tool change reset signal TLRST from the PMC or operating on the MDI.

> ⚠ **CAUTION**
> If the tool life is specified by use count, the tool change signal TLCH becomes 1 when the CNC is reset by a command such as M02 or M30 after the tool life has expired or when the tool life count restart M code is issued.
> If the tool life count type is specified by duration, the TLCH becomes 1 when the tool life expires, even if machining is in progress. Machining, however, continues until the end of the program.

Tool change reset signal TLRST <Gn048.7>

[Classification] Input signal

[Function] Clears all execution data, such as tool life count values and marks "*" and "@", for a group.
After replacing all tools in the groups in which the life of all the tools has expired (as shown on the screen) with new ones, input this signal by specifying a group number, using the tool group number selection signals (TL01 to TL512). Setting bit 4 (GRS) of parameter No. 6800 to 1 can clear execution data for all registered groups without inputting the tool group number select signal.
Execution data can be cleared also by operating on the MDI.

[Operation] When the signal changes from 0 to 1, the control unit behaves as explained below.

- If the lives of all tools in the group specified by the tool group number selection signals (TL01 to TL512) have expired, the related information is cleared. Therefore, when a subsequent program command specifies the group number, tools are selected from the beginning. If the group specified by the tool group number selection signals (TL01 to TL512) contains at least one tool whose life has not expired, no special operation takes place.

> **NOTE**
> Set the tool change reset signal TLRST from 0 to 1 only when the control unit is not in the reset state (the RST signal is 1) and the following conditions are satisfied:
> 1) If bit 5 (TRS) of parameter No. 6805 = 0
> Reset state (the OP signal is 0)
> 2) If bit 5 (TRS) of parameter No. 6805 = 1
> <1>Reset state (the OP signal is 0)
> <2>Automatic operation stop state (the STL signal and the SPL signal are 0 and the OP signal is 1)
> <3>Automatic operation stop state (the STL signal is 0 and the SPL signal is 1)
> (In the automatic operation stop state, automatic operation pause state, and automatic operation start state (the STL signal is 1) during execution of a data setting command (G10 L3), however, the tool change reset signal TLRST is invalid.)

Individual tool change signal TLCHI <Fn064.2>

[Classification] Output signal

[Function] When the tool life count is specified by duration, this signal informs that the life of the currently used tool has expired. The signal can be used to cause an interrupt to program execution, to run a tool change program, and, after tool exchange, to resume the interrupted program execution.

[Operation] The signal becomes 1 when:

- The life of the currently used tool has expired.

The signal becomes 0 when:

- An individual tool change reset is executed.

Individual tool change reset signal TLRSTI <Gn048.6>

[Classification] Input signal

[Function] Resets the individual tool change signal TLCHI to 0.

[Operation] When this signal becomes 1, the control unit behaves as explained below.

- The individual tool change signal is reset to 0.

> **NOTE**
> 1 These signals are valid only for tool life management in which the tool life count type is specification by duration.
> 2 The individual tool change signal TLCHI is not cleared by a reset.

Tool skip signal TLSKP <Gn048.5>

[Classification] Input signal

[Function] Skips a tool whose life has not expired and selects the next tool forcibly. One of the following two methods can be selected according to the setting of bit 3 (SIG) of parameter No. 6800:

(i) How to specify a group number by using the tool group number select signals (SIG = 1)
Specify the group number of the tool to be skipped by using the tool group number selection signals (TL01 to TL512), then set the tool skip signal to 1. The next T code command will then select the next tool in the group which is specified to be skipped.

(ii) How not to specify a group number by using the tool group number select signals (SIG = 0)
Set the tool skip signal TLSKP to 1 without specifying any group number. The group to which the currently selected tool belongs is then assumed to be specified.
The next T code command will select the next tool in the group for which the skip is specified.

When the tool skip signal TLSKP is set to 1 for the last tool, the tool change signal TLCH is set to 1.

[Operation] When this signal is set from 0 to 1, the control unit behaves as explained below:

- Among the tools whose lives have not expired in the group for which a skip operation is specified, the smallest tool number in the tool life management table is marked with "#". When the group is specified again by a T code command, the marked tool is skipped, and the next new tool is selected.
  When the tool skip signal TLSKP is set to 1 for the last tool, the tool change signal TLCH is set to 1.

⚠ **CAUTION**

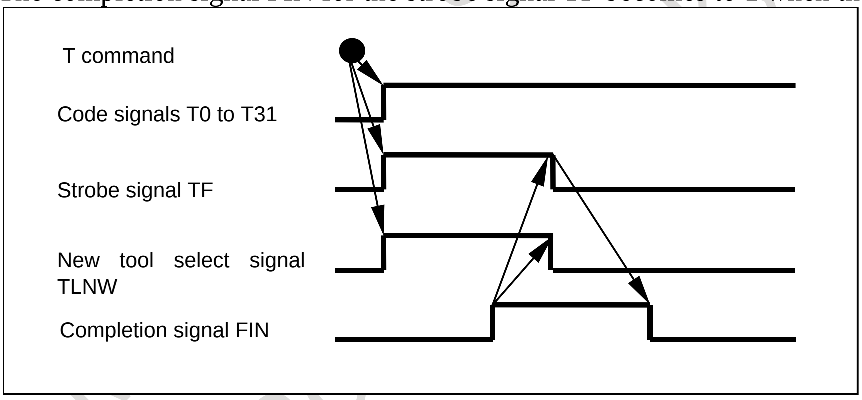
The tool skip signal TLSKP can be used even in the automatic operation start state (the STL signal is 1), but a tool selection from a group by a T command is made when the command is buffered. This means that after the command is buffered, inputting the tool skip signal TLSKP does not select the next tool. Therefore, when supplying the tool skip signal TLSKP in the automatic operation start state (the STL signal is 1), disable tool selection, for example, by suppressing buffering before setting the signal from 0 to 1.

New tool select signal TLNW <Fn064.1>

[Classification] Output signal

[Function] Notifies the PMC that a new tool has been selected from a group. This signal can be used, for example, if the tool length offset value of a newly selected tool is to be measured automatically when the tool is selected.

[Output cond.] The signal becomes 1 if:
A next new tool is selected because the life of the tools used in the group has expired when a tool group number is commanded by T code command.
After the code signal for the new tool is sent out, the TLNW becomes 1 at the same time when the tool function strobe signal TF is sent out.
The signal becomes 0 if:
The completion signal FIN for the strobe signal TF becomes to 1 when the TLNW is 1.



Tool group number selection signals TL01 to TL512 <Gn047.0 to Gn048.1>

[Classification] Input signal

[Function] These signals specify a tool group number. Before the tool change reset signal TLRST or the tool skip signal TLSKP is input, the target group for the tool change reset operation or tool skip operation is specified with these signals.

[Operation] The table given below shows the correspondence between tool group numbers and the tool group signals by providing several examples. A binary number plus 1 indicates a tool group number. A group with a specified number is then selected.

TL512	TL256	TL128	TL64	TL32	TL16	TL08	TL04	TL02	TL01	Tool group number
0	0	0	0	0	0	0	0	0	0	1
0	0	0	0	0	0	0	0	0	1	2
0	0	0	0	0	0	0	0	1	0	3
0	0	0	0	0	0	0	0	1	1	4
0	0	0	0	0	0	0	1	0	0	5
0	0	0	0	0	0	1	0	0	1	10
0	0	0	0	0	0	1	1	1	0	15
0	0	0	0	0	1	0	0	1	1	20

TL512	TL256	TL128	TL64	TL32	TL16	TL08	TL04	TL02	TL01	Tool group number
0	0	0	0	0	1	1	1	0	1	30
0	0	0	0	1	0	0	1	1	1	40
0	0	0	0	1	1	0	0	0	1	50
0	0	0	1	1	0	0	0	1	1	100
0	0	0	1	1	1	1	1	1	0	127
0	0	0	1	1	1	1	1	1	1	128
0	0	1	1	0	0	0	1	1	1	200
0	0	1	1	1	1	1	1	1	1	256
0	1	0	0	0	0	0	0	0	0	257
0	1	0	0	1	0	1	0	1	1	300
0	1	1	0	0	0	1	1	1	1	400
0	1	1	1	1	1	0	0	1	1	500
0	1	1	1	1	1	1	1	1	1	512
1	0	0	0	0	0	0	0	0	0	513
1	0	0	1	0	1	0	1	1	1	600
1	0	1	0	1	1	1	0	1	1	700
1	1	0	0	0	1	1	1	1	1	800
1	1	1	0	0	0	0	0	1	1	900
1	1	1	1	1	0	0	1	1	1	1000
1	1	1	1	1	1	1	1	1	1	1024

Tool life count override signals *TLV0 to *TLV9 <Gn049.0 to Gn050.1>

[Classification] Input signal

[Function] If the tool life count type is specified by duration, the life count can be overridden by setting bit 2 (LFV) of parameter No. 6800 = 1. Ten binary code signals are available, and they correspond to override values as follows:

$$\text{Override value} = \sum_{i=0}^{9} \{2i \times Vi\} \text{ times}$$

Keep the override values within the range stated above.
Where, if *TLVi is 1, Vi = 0
if *TLVi is 0, Vi = 1
Therefore, each signal has the following weight:

Signal	Magnification	Signal	Magnification	Signal	Magnification
*TLV0	0.1	*TLV4	1.6	*TLV8	25.6
*TLV1	0.2	*TLV5	3.2	*TLV9	51.2
*TLV2	0.4	*TLV6	6.4	-	-
*TLV3	0.8	*TLV7	12.8	-	-

Example:
If *TLV7, *TLV6, and *TLV3 are 0, the override value is calculated as follows: 12.8 + 6.4 + 0.8 = 20.0
So, the life count is multiplied by 20.
If all signals are 1, the override value is 0. So, set an override value within the range from 0 to 99.9 in steps of 0.1.
If 99.9 times is exceeded, the actual override value is clamped by 99.9 times.

[Operation] The actual cutting time obtained by life counting specified by duration is multiplied by the override value specified by these signals, and the multiplication result is assumed to be the count time for tool life management. For example, let the override value be 0.1, and the actual cutting time be 1000 seconds. Then, the tool life count time is assumed to be 100 seconds.

Tool life arrival notice signal TLCHB <Fn064.3>

[Classification] Output signal

[Function] If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1), specifying a remaining life value to be used till selection of a new tool enables the tool life arrival notice signal to be issued when life counting shows that the remaining life value of a group (life value minus life counter value) becomes lower than or equal to the remaining life setting, in order to inform in advance that the tool life will expire soon.

By bit 4 (ARL) of parameter No. 6802, it is possible to select whether to output the tool life arrival notice signal for each tool or the last tool in the group.

Bit 3 (GRP) of parameter No. 6802 can be used to specify which remaining life setting (that is, one specified in parameter Nos. 6844 and 6855 or one registered as tool life management data) to use.

[Output cond.] The signal is set to 1 if:

- Remaining life (life value - life counter value) ≤ remaining life setting provided that bit 7 (RMT) of parameter No. 6802 = 0
- Remaining life (life value - life counter value) = remaining life setting provided that bit 7 (RMT) of parameter No. 6802 = 1

The signal is reset to 0 if:

- Remaining life (life value - life counter value) > remaining life setting provided that bit 7 (RMT) of parameter No. 6802 = 0
- Remaining life (life value - life counter value) ≠ remaining life setting provided that bit 7 (RMT) of parameter No. 6802 = 1

> **NOTE**
> 1 The signal changes when life counting is performed.
> 2 When using a life count override, reset bit 7 (RMT) of parameter No. 6802 to 0.
> 3 When the life count is specified by duration, the unit of the remaining life and remaining life setting to be compared varies depending on the life count interval (bit 0 (FCO) of parameter No. 6805). If the tool life is counted every second, the unit of the values to be compared is one minute; if the tool life is counted every 0.1 second, the unit of the values to be compared is 0.1 minute.

If bit 7 (CTB) of parameter No. 6803 is 1, this signal can be set to 0 when an operation such as the following is performed.

- Clears the execution data on the list screen of tool life management.
- Deletes all tool group data at a time, adds a tool number, or deletes tool data on the edit screen of tool life management.
- Clears the execution data by the tool change reset signal TLRST <Gn048.7>.
- Registers, changes, or deletes all tool life management group data by the G10 command.
- Executes the FOCAS2 cnc_clrcntinfo function (which clears the tool life counter or tool information).
- Replaces a tool with a tool of which life is not managed by the M06 command.

Tool life counting disable signal LFCIV<Gn048.2>

[Classification] Input signal

[Function] Disables tool life counting for selected tools.

[Operation] If the signal turns to 1, the control unit behaves as explained below.

- No life counting is performed for selected tools.

NOTE

The tool life counting disable signal LFCIV<Gn048.2> is valid if bit 6 (LFI) of parameter No. 6804 is 1.

Tool life counting disabled signal LFCIF<Fn093.2>

[Classification] Output signal

[Function] Informs that tool life counting is disabled for selected tools.

[Output cond.] The signal becomes 1 when:

- Tool life counting is disabled because the tool life counting disable signal LFCIV is 1.

The signal becomes 0 when:

- Tool life counting is enabled because the tool life counting disable signal LFCIV is 0.

Number of remaining tools notification signal TLAL <Fn154.0>

[Classification] Output signal

[Function] Informs that the number of remaining tools in the group selected with a T code command is equal to or less than the setting of parameter No. 6846.

[Output cond.] The signal becomes 1 when:

- The number of remaining tools in the group selected with a T code command is equal to or less than the setting of parameter No. 6846.

The signal becomes 0 when:

- A value is input to parameter No. 6846.
- Using a G10 command, registration is performed that involves the erasure of all life management data groups.
- Using a T code command, a group is selected in which the number of remaining tools is greater than the setting of parameter No. 6846.

The signal also becomes 0 when any of the following is executed in the group that has caused the number of remaining tools notification signal TLAL to be generated.

- Clearing of execution data, using the tool change reset signal TLRST <Gn048.7>.
- Change and deletion of life management data, using a G10 command.
- Batch deletion of tool groups from the group edit screen of tool life management, addition of tool data to the screen, and deletion from the screen.
- Clearing of tool execution data from the group edit screen of tool life management.
- Clearing of execution data from the list screen of tool life management.
- Execution of any of the FOCAS2 functions below.
  (cnc_clrcntinfo, cnc_deltlifegrp, cnc_deltlifedt, cnc_instlifedt, cnc_wrt1info, cnc_wrt2info, cnc_wr1tlifedata, cnc_wr1tlifedat2, cnc_wr2tlifedata)

NOTE

1 This signal is used in the M series only.

2 If the setting of parameter No. 6846 is 0, the number of remaining tools notification signal TLAL is not output.

Example)

If three tools are registered in a group, and the number of remaining tools is set to 1 (parameter No. 6846 = 1), the timing chart of the number of remaining tools notification signal TLAL is as shown in Fig. 11.5 (c).

The number of remaining tools notification signal TLAL is output at the same time as the T code for the third tool.

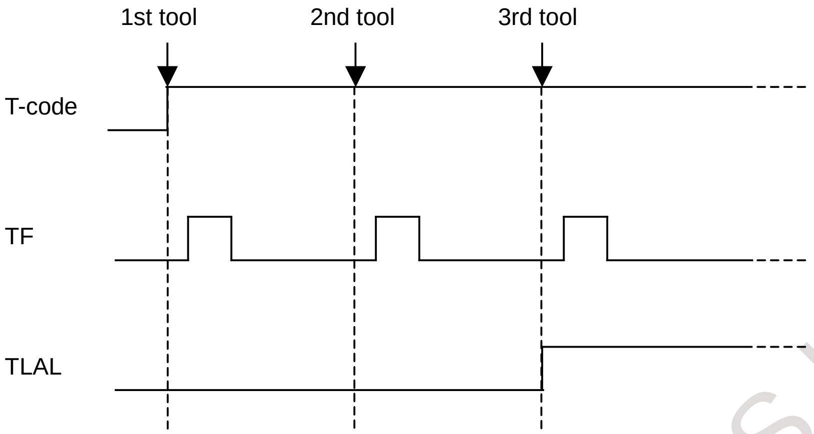


Fig. 11.5 (c) Timing chart of the number of remaining tools notification signal TLAL

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0046		**KEY4**	**KEY3**	**KEY2**	**KEY1**			
Gn047	**TL128**	**TL64**	**TL32**	**TL16**	**TL08**	**TL04**	**TL02**	**TL01**
Gn048	**TLRST**	**TLRSTI**	**TLSKP**			**LFCIV**	**TL512**	**TL256**
Gn049	***TLV7**	***TLV6**	***TLV5**	***TLV4**	***TLV3**	***TLV2**	***TLV1**	***TLV0**
Gn050							***TLV9**	***TLV8**

	#7	#6	#5	#4	#3	#2	#1	#0
Fn064					**TLCHB**	**TLCHI**	**TLNW**	**TLCH**
Fn093						**LFCIF**		
Fn154								**TLAL**

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6800	**M6T**	**IGI**	**SNG**	**GRS**	**SIG**	**LTM**	**GS2**	**GS1**

[Input type] Parameter input
[Data type] Bit path

#0 GS1
#1 GS2 For the maximum number of groups set in parameter No. 6813, up to four tools can be registered per group. The combination of the number of registrable groups and the number of tools per group can be changed by setting GS1 and GS2.

GS2	GS1	Number of groups	Number of tools
0	0	1/8 of maximum number of groups (No. 6813)	32
0	1	1/4 of maximum number of groups (No. 6813)	16
1	0	1/2 of maximum number of groups (No. 6813)	8
1	1	Maximum number of groups (No. 6813)	4

NOTE
After changing these parameters, set data again by using G10 L3 ;(registration with deleting data of all groups).

#2 LTM The tool life count type is specified by:
0: Count.
1: Duration.

NOTE
After changing this parameter, set data again by using G10 L3 ;(registration after deletion of data of all groups).

#3 SIG In tool skip by signals, the group number is:
0: Not input by the tool group number selection signals TL01～TL512<Gn047.0 ～ Gn048.1>.
1: Input by the tool group number selection signals.

NOTE
When this parameter is set to 0, a tool of the currently used group is skipped.

#4 GRS When the tool change reset signal TLRST <Gn048.7> is input:
0: If the life of the group specified by the tool group number selection signals TL01 to TL512 <Gn047.0 to Gn048.1> has expired, the execution data of the group is cleared.
1: The execution data of all registered groups is cleared.
If this parameter is set to 1, the execution data of all registered groups are cleared also when the clear operation to clear execution data is performed on the tool life management list screen.

#5 SNG When the tool skip signal TLSKP <Gn048.5> is input while a tool not controlled by the tool life management function is being used:
0: A tool of the most recently used group or a specified group (bit 3 (SIG) of parameter No. 6800) is skipped.
1: The tool skip signal is ignored.

#6 IGI Tool back numbers are:
0: Not ignored.
1: Ignored.

#7 M6T A T code specified in the same block as M06 is:
0: Assumed to be a back number.
1: Assumed to be a command specifying the next tool group.

	#7	#6	#5	#4	#3	#2	#1	#0
6801	M6E				EMD	LVF	TSM	
	M6E				EMD	LVF		

NOTE
The use of this parameter varies depending on whether the tool management function or tool life management function is used.

[Input type] Parameter input

[Data type] Bit path

#1 TSM In the tool life management function, life counting is performed as follows when more than one offset is specified:
0: Counting is performed for each tool number.
1: Counting is performed for each tool.

#2 LVF When the life value is counted by duration in the tool management function or tool life management function, tool life count override signals *TLV0 to *TLV9 <Gn049.0 to Gn050.1> are:
0: Not used.
1: Used.

#3 EMD In the tool life management function, the mark "*" indicating that the life has expired is displayed when:
0: The next tool is used.
1: The life has just expired.

> **NOTE**
> If this parameter is set to 0, the "@" mark (indicating that the tool is in use) is kept displayed unless the next tool whose life has not expired is used. If this parameter is set to 1, marks are displayed in different ways depending on the life count type.
> If the life count type is the duration specification type, the "*" mark (indicating that the life has expired) appears when the life has expired. If the life count type is the count specification type, one count is not assumed until the end of the program (M02, M30, and so on). Therefore, even when the life value and the tool life counter value match, the "*" mark (life has expired) does not appear. The "*" mark (life has expired) appears when the tool is used again by a tool group command (T code) or tool change command (M06) issued after the CNC is reset.

#7 M6E When a T code is specified in the same block as M06:
0: The T code is treated as a back number or the group number to be selected next. Which number is assumed depends on the setting of bit 7 (M6T) of parameter No. 6800.
1: Life counting for the tool group starts immediately.

	#7	#6	#5	#4	#3	#2	#1	#0
6802	RMT	TSK	TGN	ARL	GRP	E17	TCO	T99

[Input type] Parameter input
[Data type] Bit path

#0 T99 When M99 of the main program is executed, and if there is a tool group whose life is expired:
0: The tool change signal is not output.
1: The tool change signal is output and the automatic operation stop state is entered..

If the life count is specified by use count and this parameter 1, the tool change signal TLCH <Fn064.0> is output and the automatic operation becomes a stopped state if there is a tool group whose life is expired when the M99 command is specified.

If the life count type is the duration specification type, the automatic operation becomes a stopped state if the life of at least one tool group has expired.

M

If the life count is specified by use count, after the M99 command is specified, a tool group command (T code) selects, from a specified group, a tool whose life has not expired, and the next tool change command (M06) increments the tool life counter by one.

T

If the life count is specified by use count, when a tool group command (T code) is specified after the M99 command is specified, a tool whose life has not expired is selected from a specified group, and the tool life counter is incremented by one.
When the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), the same specifications as for the M series apply.

#1 TCO
#2 E17 Specifies whether to allow the FOCAS2 or PMC window function to write tool information of a group being used or a group to be used next during automatic operation (the automatic operation signal OP <Fn000.7> is set to 1).

<table>
<tr><th colspan="3" rowspan="4">Condition</th><th colspan="3">Bit 1 (TCO) of parameter No. 6802</th></tr>
<tr><th rowspan="3">0</th><th colspan="2">1</th></tr>
<tr><th colspan="2">Bit 2 (E17) of parameter No. 6802</th></tr>
<tr><th>1</th><th>0</th></tr>
<tr><td rowspan="3">During automatic operation</td><td rowspan="2">Group being used or to be used next</td><td>Tool being used</td><td>×</td><td>△</td><td>○</td></tr>
<tr><td>Tool not being used</td><td>×</td><td>○</td><td>○</td></tr>
<tr><td colspan="2">Group neither being used nor to be used next</td><td>○</td><td>○</td><td>○</td></tr>
<tr><td colspan="3">Not during automatic operation</td><td>○</td><td>○</td><td>○</td></tr>
</table>

○: Tool information can be written from FOCAS2 and PMC window.
×: Tool information cannot be written from FOCAS2 and PMC window.
When an attempt is made to write tool information from PMC window, completion code 13 (REJECT ALARM) is returned.
△: Tool information cannot be cleared.

NOTE
When tool information of a tool being used (marked with "@") in the group being used or to be used next or tool information of the most recently used tool (marked with "@") in a group that is neither the group being used nor the group to be used next is cleared, the life counter is reset to 0.
It is possible to modify tool information of a tool in the group to be used next. However, because tool selection is already completed, the selected tool does not change even when the tool information is modified.
This parameter has no influence on modifications to tool information by edit operations from the tool life management screen.

#3 GRP Management data of tool life arrival notice signal TLCHB <Fn064.3> is:
0: Managed using the remaining life value set in parameter No. 6844 and 6845.
1: Managed using the remaining life value set in tool life management data.

> **NOTE**
> When the tool life arrival notice signal function is used, bit 4 (LFB) of parameter No. 6805 must be set to 1 to enable the tool life management B function.

#4 ARL Tool life arrival notice signal TLCHB <Fn064.3> of tool life management is:
- 0: Output for each tool.
- 1: Output for the last tool of a group.

This parameter is valid only when bit 3 (GRP) of parameter No. 6802 is set to 1.

#5 TGN In the tool life management function, the optional group number function is:
- 0: Not used.
- 1: Used.

> **NOTE**
> When the optional group number function is used, bit 4 (LFB) of parameter No. 6805 must be set to 1 to enable the tool life management B function.
> In lathe systems, the optional group number function can be used if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1).

#6 TSK If the count type in tool life management is the duration type, then when the last tool of a group is skipped by a signal:
- 0: The count value for the last tool equals the life value.
- 1: The count value for the last tool remains unchanged.

#7 RMT Tool life arrival notice signal TLCHB <Fn064.3> is turned 1 and 0 as follows:
- 0: The signal is turned 1 if the remaining life value (the life value minus the life counter value) is smaller than or equal to the remaining life setting. The signal is turned 0 if the remaining life value (the life value minus the life counter value) is greater than the remaining life setting.
- 1: The signal is turned 1 if the remaining life value (the life value minus the life counter value) is equal to the remaining life setting. The signal is turned 0 if the remaining life value (the life value minus the life counter value) is not equal to the remaining life setting.

> **NOTE**
> When using the life count override feature, set bit 7 (RMT) of parameter No. 6802 to 0. When the life count is specified by duration, the unit used for determining the result of comparison between the remaining life and the remaining life setting varies depending on the life count interval (bit 0 (FCO) of parameter No. 6805). If the life is counted every second, the comparison is made in units of 1 minute; if the life is counted every 0.1 second, the comparison is made in units of 0.1 minute.

	#7	#6	#5	#4	#3	#2	#1	#0
6803	CTB							

[Input type] Parameter input

[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off once.

 # 7 CTB Whether to turn the tool life arrival notice signal TLCHB <Fn064.3> of tool life management off is determined when life counting starts. An additional turn-off condition is:
0: Not added.
1: Added.
The tool life arrival notice signal is turned off when one of the following operations is performed for the currently used group:
- Clears the execution data on the list screen of tool life management.
- Deletes all tool group data at a time, adds a tool number, or deletes tool data on the edit screen of tool life management.
- Clears the execution data by the tool change reset signal TLRST <Gn048.7>.
- Registers, changes, or deletes all tool life management group data by the G10 command.
- Executes the FOCAS2 cnc_clrcntinfo function (which clears the tool life counter or tool information).
- Replaces a tool with a tool of which life is not managed by the M06 command.

	#7	#6	#5	#4	#3	#2	#1	#0
6804		LFI				ETE	TCI	

[Input type] Parameter input
[Data type] Bit path

#1 TCI During automatic operation (the automatic operation signal OP<Fn000.7> is 1), editing of tool life data is:
0: Disabled.
1: Enabled.

> **NOTE**
> When this parameter is set to 1, tool life data can be edited even during automatic operation (the OP signal is 1). If the target group for editing is the group being used or the group to be used next, however, only presetting of the life counter is permitted, and other data cannot be modified.

#2 ETE In the tool life management screen, when the life of the last tool in the group is expired, the mark of the tool:
0: depends on setting bit 3 (EMD) of parameter No. 6801.
1: is "*" mark.

If bit 2 (ETE) of parameter No. 6804 is set to 1, when the life counter of the final tool in the group becomes equal to the life value, the mark "*" indicating that the life of tool has been expired is displayed on the last tool of the tool life management screen.
When tool change signal TLCH <Fn064.0> is 1, if the tool information of the final tool is acquired in FOCAS 2 or PMC window, the tool indicates that the life has been expired.

#6 LFI In tool life management, counting of the life of a selected tool is:
0: Enabled.

1: Enabled or disabled according to the status of tool life counting disable signal LFCIV <Gn048.2>.

	#7	#6	#5	#4	#3	#2	#1	#0
6805	TAD	TRU	TRS	LFB			FGL	FCO

[Input type] Parameter input
[Data type] Bit path

#0 FCO If the life count type is the duration specification type, the life is counted as follows:
0: Every second.
1: Every 0.1 second.
According to the setting of this parameter, the increment system of life values and tool life counter values displayed on the tool life management screen is set as follows:

Parameter FCO	0	1
Increment system for display and setting of life values and life counter values	1-minute increments	0.1-minute increments

NOTE
After changing the setting of this parameter, set data again by using G10L3;(registration after deletion of data of all groups).

#1 FGL If the life count type is the duration specification type, life data registered by G10 is:
0: In minute increments.
1: In 0.1-second increments.

#4 LFB The tool life management B function is:
0: Disabled.
1: Enabled.
When the tool life management B function is enabled, the following functions can be used:
<1> Tool life value extension (count specification: 99999999 times, duration specification: 100000 minutes)
<2> Optional group number function
<3> Tool life arrival notice function
In lathe systems, if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), the optional group number function can be used.

#5 TRS Tool change reset signal TLRST <Gn048.7> is valid when reset signal RST <Fn001.1> is not 1 and:
0: The reset state (automatic operation signal OP<Fn000.7> is 0) is observed.
1: The reset state (automatic operation signal OP is 0), automatic operation stop state (cycle start lamp signal STL <Fn000.5> and feed hold lamp signal SPL <Fn000.4> are 0 and OP is 1), or the automatic operation pause state (STL is 0 and SPL is 1) is observed. The TLRST signal, however, is invalid when the automatic operation stop state, automatic operation pause state, and automatic operation start state (STL is 1) is observed during execution of a data setting command (G10L3).

#6 TRU When the life count type is the duration specification type, and the life is counted every second (bit 0 (FCO) of parameter No. 6805 is set to 0):
0: Cutting time less than one second is discarded and is not counted.
1: Cutting time less than one second is rounded up and is counted as one second.

> **NOTE**
> If the life is counted every 0.1 second (bit 0 (FCO) of parameter No. 6805 is set to 1), cutting time less than 0.1 second is always rounded up and is counted as 0.1 second.

#7 TAD With tool change type D (bit 7 (M6E) of parameter No. 6801 is set to 1), when a block specifying M06 contains no T command:
0: An alarm PS0153, "T-CODE NOT FOUND" is issued.
1: No alarm is issued.

6810	Tool life management ignore number

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999999

This parameter sets the tool life management ignore number.
When the value specified in a T code exceeds the value set in this parameter, the value obtained by subtracting the parameter-set value from the T code value is assumed to be the tool group number for tool life management.

6811	Tool life count restart M code

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 127 (except 01, 02, 30, 98, and 99)

When 0 is specified, it is ignored.
When the life is specified by count, the tool change signal TLCH <Fn064.0> is output if the life of at least one tool group has expired when the tool life count restart M code is issued.
The T code (tool life management group command) specified after the tool life count restart M code selects a tool whose life has not expired from a specified group, and the next M06 command increments the tool life counter by one.
When the life is specified by duration, specifying the tool life count restart M code causes nothing. When 0 is set in this parameter, the tool life count restart M code is invalid. When the data of M code exceeds 127 values, set 0 in parameter No. 6811, and set the value of M code in parameter No. 13221. The data range of parameter No. 13221 is from 0 to 255.

> **NOTE**
> The use of this parameter varies depending on whether the tool management function or tool life management function is used.

6813	Maximum number of groups in tool life management

> **NOTE**
> After this parameter has been set, the power must be turned off once.

[Input type] Parameter input
[Data type] Word path
[Unit of data] Group
[Valid data range] 0, 8, 16 to 256

This parameter sets the maximum number of groups to be used for each path. As the maximum number of groups, set a multiple of eight. When the tool life management function is not used, 0 must be set. Set this parameter so that the total number of groups in all paths does not exceed the total number of groups in the entire system (256 groups).

> **NOTE**
> When the power is turned on, all tool life management file data is initialized. So, tool life management data must be set for all paths that use tool life management.

6844	Remaining tool life (use count)

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 9999

This parameter sets a remaining tool life (use count) used to output the tool life expiration notice signal TLCHB <Fn064.3> when the tool life is specified by use count. If a value greater than the tool life value or 0 is set in this parameter, the tool life notice signal is not output.

6845	Remaining tool life (use duration)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min
[Valid data range] Not greater than the tool life value

This parameter sets the remaining tool life (use duration) used to output the tool life notice signal TLCHB <Fn064.3> when the tool life is specified by use duration. If a value greater than the tool life value or 0 is specified in this parameter, the tool life notice signal is not output.

> **NOTE**
> When the life is counted every 0.1 second (bit 0 (FCO) of parameter No. 6805 = 1), the parameter value is in 0.1-minute increments.

6846	Remaining tool number in a group

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 127

This parameter sets the remaining tool number in a group.
When the remaining tool number in the group selected by the T code is smaller than or equal to the value set in this parameter, the remaining tool number notice signal TLAL <Fn154.0> is output. When this parameter is set to 0, the remaining tool number notice signal is not output.

13221	M code for tool life count restart

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 255 (not including 01, 02, 30, 98, and 99)

When 0 is set, this parameter is ignored.

For the operation of an M code for tool life count restart, see the description of parameter No. 6811.
This parameter is used when an M code for tool life count restart exceeds 127.
Set parameter No. 6811 to 0, and set the value of an M code in this parameter.

> **NOTE**
> The use of this parameter varies depending on whether it is used by the tool management function or tool life management function.

13265	H code for using the tool length offset in tool life management

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 9999

Usually, when H99 is specified, tool length offset is enabled by the H code of the tool being used. By setting any H code in this parameter, any H code instead of H99 can be used. If 0 is specified, H99 is assumed.
A value ranging from 0 to 9999 can be set.

> **NOTE**
> The use of this parameter varies depending on whether it is used by the tool management function or tool life management function.

13266	D code for enabling cutter compensation in tool life management

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 9999

Usually in tool life management, specifying D99 allows the D code of the tool being used to enable cutter compensation. By setting any D code in this parameter, the D code instead of D99 can be used. If 0 is set, D99 is assumed.
The valid data range is 0 to 9999.

Alarm and message

Number	Message	Description
PS0149	FORMAT ERROR IN G10L3	In registration (G10L3 to G11) of tool life management data, an address other than Q1, Q2, P1, and P2 or an unusable address was specified.
PS0150	ILLEGAL LIFE GROUP NUMBER	The tool group number exceeded the maximum allowable value. The tool group number (P after specification of G10 L3;) or the group number given by the tool life management T code in a machining program is exceeded the maximum allowable value.
PS0151	GROUP NOT FOUND AT LIFE DATA	The tool group specified in a machining program is not set in tool life management data.
PS0152	OVER MAXIMUM TOOL NUMBER	The number of tools registered in one group exceeded the maximum allowable registration tool number.
PS0153	T-CODE NOT FOUND	In registration of tool life data, a block in which the T code needs to be specified does not include the T code. Alternatively, in tool exchange method D, M06 is specified solely. Modify the program.
PS0154	NOT USING TOOL IN LIFE GROUP	The H99 command, D99 command, or the H/D code set by parameters Nos. 13265 and 13266 was specified when no tool belonging to a group is used.
PS0155	ILLEGAL T-CODE COMMAND	In the machining program, the T code that is present in the block containing M06 does not correspond to the group currently being used. Modify the program.

Number	Message	Description
PS0156	P/L COMMAND NOT FOUND	The P and L commands are not specified in the beginning of a program for setting a tool group. Modify the program.
PS0157	TOO MANY TOOL GROUPS	In registration of tool life management data, the group setting command block counts of P (group number) and L (tool life) exceeded the maximum group count.
PS0158	TOOL LIFE VALUE OUT OF RANGE	The life value that is being set is too large. Change the setting.
PS0159	ILLEGAL TOOL LIFE DATA	Tool life management data is corrupted for some reason. Register the tool data in the tool group or the tool data in the group again by G10L3; or MDI input.
PS0430	TOOL LIFE PAIRS ZERO	Tool life management group number parameter No. 6813 is 0.
PS0431	ILLEGAL T/R DATA OF TOOL LIFE	The arbitrary group number (T) or remaining amount setting (R) is invalid.
IO1104	OVER MAXIMUM TOOL LIFE PAIRS	The maximum number of tool life management pairs is exceeded. Modify the setting of the maximum number of tool life management pairs in parameter No. 6813.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Tool life management

12 PROGRAM COMMAND

12.1 DECIMAL POINT PROGRAMMING / CALCULATOR-TYPE DECIMAL POINT PROGRAMMING

Overview

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:

X, Y, Z, U, V, W, A, B, C, I, J, K, Q, R, F ......Machining center system (for a type common to all axes)
X, Y, Z, U, V, W, A, B, C, I, J, K, R, F ....Lathe system

There are two types of decimal point notation: calculator-type notation and standard notation.
When calculator-type decimal point notation is used, a value without decimal point is considered to be specified in millimeters, inches or degree. When standard decimal point notation is used, such a value is considered to be specified in least input increments. Select either calculator-type or standard decimal point notation by using the bit 0 (DPI) of parameter No. 3401. Setting the bit 0 (AXDx) of parameter No. 3455 to 1 enables the calculator-type decimal input to be set up for individual axes separately. Values can be specified both with and without decimal point in a single program.

Program command	Calculator-type decimal point programming	Standard type decimal point programming
X1000 Command value without decimal point	1000mm Unit : mm	1mm Unit : Least input increment (0.001mm)
X1000.0 Command value with decimal point	1000mm Unit : mm	1000mm Unit : mm

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3401								DPI

[Input type] Parameter input
[Data type] Bit path

#0 DPI When a decimal point is omitted in an address that can include a decimal point
0: The least input increment is assumed. (Normal decimal point input)
1: The unit of mm, inch, degree, or second is assumed. (Calculator- type decimal point input)

	#7	#6	#5	#4	#3	#2	#1	#0
3455								AXDx

[Input type] Parameter input
[Data type] Bit axis

#0 AXDx If a decimal point is omitted for an axis address with which a decimal point can be used, the value is determined:
0: In accordance with the least input increment. (Normal decimal point input)
1: In millimeters, inches, degrees, or seconds. (calculator-type decimal point input)

NOTE
This parameter specifies the calculator-type decimal point input function for each axis.
For the same axis name, be sure to make the same setting.

Alarm and message

Number	Message	Description
PS0007	ILLEGAL USE OF DECIMAL POINT	A decimal point (.) was specified at an address where no decimal point may be specified, or two decimal points were specified.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Decimal point programming

12.2 G CODE SYSTEM

12.2.1 G Code List in the Lathe System

T

Overview

There are three G code systems : A,B, and C. Select a G code system using bits 7 (GSC) and 6 (GSB) of parameter No. 3401.

Table 12.2.1 G code list

G code system			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01	01	Linear interpolation (Cutting feed)
G02	G02	G02	01	Circular interpolation CW or helical interpolation CW
G03	G03	G03	01	Circular interpolation CCW or helical interpolation CCW
G04	G04	G04	00	Dwell
G05	G05	G05	00	AI contour control (command compatible with high precision contour control)
G05.1	G05.1	G05.1	00	AI contour control
G05.4	G05.4	G05.4	00	HRV3 on/off
G07.1 (G107)	G07.1 (G107)	G07.1 (G107)	00	Cylindrical interpolation
G08	G08	G08	00	AI contour control (advanced preview control compatible command)
G09	G09	G09	00	Exact stop
G10	G10	G10	00	Programmable data input
G10.6	G10.6	G10.6	00	Tool retract and recover
G10.8	G10.8	G10.8	00	Programmable internal data change
G11	G11	G11	00	Programmable data input cancel
G12.1 (G112)	G12.1 (G112)	G12.1 (G112)	21	Polar coordinate interpolation mode
G13.1 (G113)	G13.1 (G113)	G13.1 (G113)	21	Polar coordinate interpolation cancel mode
G17	G17	G17	16	XpYp plane selection
G18	G18	G18	16	ZpXp plane selection

G code system			Group	Function
A	B	C		
G19	G19	G19	16	YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71	06	Input in mm
G22	G22	G22	09	Stored stroke check function on
G23	G23	G23	09	Stored stroke check function off
G25	G25	G25	08	Spindle speed fluctuation detection off
G26	G26	G26	08	Spindle speed fluctuation detection on
G27	G27	G27	00	Reference position return check
G28	G28	G28	00	Return to reference position
G28.2	G28.2	G28.2	00	In-position check disable reference position return
G29	G29	G29	00	Movement from reference position
G30	G30	G30	00	2nd, 3rd and 4th reference position return
G30.2	G30.2	G30.2	00	In-position check disable 2nd, 3rd, or 4th reference position return
G31	G31	G31	00	Skip function
G32	G33	G33	01	Threading
G34	G34	G34	01	Variable lead threading
G35	G35	G35	01	Circular threading CW
G36	G36	G36	01	Circular threading CCW (When bit 3 (G36) of parameter No. 3405 is set to 1) or Automatic tool offset (X axis) (When bit 3 (G36) of parameter No. 3405 is set to 0)
G37	G37	G37	01	Automatic tool offset (Z axis) (When bit 3 (G36) of parameter No. 3405 is set to 0)
G37.1	G37.1	G37.1	01	Automatic tool offset (X axis) (When bit 3 (G36) of parameter No. 3405 is set to 1)
G37.2	G37.2	G37.2	01	Automatic tool offset (Z axis) (When bit 3 (G36) of parameter No. 3405 is set to 1)
G38	G38	G38	01	Tool radius/tool nose radius compensation: with vector held
G39	G39	G39	01	Tool radius/tool nose radius compensation: corner rounding interpolation
G40	G40	G40	07	Tool radius/tool nose radius compensation : cancel
G41	G41	G41	07	Tool radius/tool nose radius compensation : left
G42	G42	G42	07	Tool radius/tool nose radius compensation : right
G40.1	G40.1	G40.1	19	Normal direction control cancel mode
G41.1	G41.1	G41.1	19	Normal direction control left on
G42.1	G42.1	G42.1	19	Normal direction control right on
G43	G43	G43	23	Tool length compensation + (Bit 3 (TCT) of parameter No. 5040 must be 1.)
G44	G44	G44	23	Tool length compensation - (Bit 3 (TCT) of parameter No. 5040 must be 1.)
G43.7 (G44.7)	G43.7 (G44.7)	G43.7 (G44.7)	23	Tool offset (Bit 3 (TCT) of parameter No. 5040 must be 1.)
G49 (G49.1)	G49 (G49.1)	G49 (G49.1)	23	Tool length compensation cancel (Bit 3 (TCT) of parameter No. 5040 must be 1.)
G50	G92	G92	00	Coordinate system setting or max spindle speed clamp
G50.3	G92.1	G92.1	00	Workpiece coordinate system preset
G50.1	G50.1	G50.1	22	Programmable mirror image cancel
G51.1	G51.1	G51.1	22	Programmable mirror image
G50.2 (G250)	G50.2 (G250)	G50.2 (G250)	20	Polygon turning cancel
G51.2 (G251)	G51.2 (G251)	G51.2 (G251)	20	Polygon turning
G50.4	G50.4	G50.4	00	Cancel synchronous control
G50.5	G50.5	G50.5	00	Cancel composite control

G code system			Group	Function
A	B	C		
G50.6	G50.6	G50.6	00	Cancel superimposed control
G51.4	G51.4	G51.4	00	Start synchronous control
G51.5	G51.5	G51.5	00	Start composite control
G51.6	G51.6	G51.6	00	Start superimposed control
G52	G52	G52	00	Local coordinate system setting
G53	G53	G53	00	Machine coordinate system setting
G53.1	G53.1	G53.1	00	Tool axis direction control
G53.2	G53.2	G53.2	00	Selecting a machine coordinate system with feedrate
G53.6	G53.6	G53.6	00	Tool center point retention type tool axis direction control
G54 (G54.1)	G54 (G54.1)	G54 (G54.1)	14	Workpiece coordinate system 1 selection
G55	G55	G55	14	Workpiece coordinate system 2 selection
G56	G56	G56	14	Workpiece coordinate system 3 selection
G57	G57	G57	14	Workpiece coordinate system 4 selection
G58	G58	G58	14	Workpiece coordinate system 5 selection
G59	G59	G59	14	Workpiece coordinate system 6 selection
G61	G61	G61	15	Exact stop mode
G63	G63	G63	15	Tapping mode
G64	G64	G64	15	Cutting mode
G65	G65	G65	00	Macro call
G66	G66	G66	12	Macro modal call A
G66.1	G66.1	G66.1	12	Macro modal call B
G67	G67	G67	12	Macro modal call A/B cancel
G68	G68	G68	04	Mirror image on for double turret or balance cutting mode
G68.1	G68.1	G68.1	17	Coordinate system rotation start or 3-dimensional coordinate system conversion mode on
G68.2	G68.2	G68.2	17	Tilted working plane indexing
G68.3	G68.3	G68.3	17	Tilted working plane indexing by tool axis direction
G68.4	G68.4	G68.4	17	Tilted working plane indexing (incremental multi-command)
G69	G69	G69	04	Mirror image off for double turret or balance cutting mode cancel
G69.1	G69.1	G69.1	17	Coordinate system rotation cancel or 3-dimensional coordinate system conversion mode off
G70	G70	G72	00	Finishing cycle
G71	G71	G73	00	Stock removal in turning
G72	G72	G74	00	Stock removal in facing
G73	G73	G75	00	Pattern repeating cycle
G74	G74	G76	00	End face peck drilling cycle
G75	G75	G77	00	Outer diameter/internal diameter drilling cycle
G76	G76	G78	00	Multiple-thread cutting cycle
G71	G71	G72	01	Traverse grinding cycle
G72	G72	G73	01	Traverse direct sizing/grinding cycle
G73	G73	G74	01	Oscillation grinding cycle
G74	G74	G75	01	Oscillation direct sizing/grinding cycle
G80	G80	G80	10	Canned cycle cancel for drilling Electronic gear box : synchronization cancellation
G80.4	G80.4	G80.4	28	Electronic gear box: synchronization cancellation
G80.5	G80.5	G80.5	27	Electronic gear box 2 pair: synchronization cancellation
G81	G81	G81	10	Spot drilling (FS15-T format) Electronic gear box : synchronization start
G81.4	G81.4	G81.4	28	Electronic gear box: synchronization start
G81.5	G81.5	G81.5	27	Electronic gear box 2 pair: synchronization start
G82	G82	G82	10	Counter boring (FS15-T format)

G code system			Group	Function
A	B	C		
G83	G83	G83	10	Cycle for face drilling
G83.1	G83.1	G83.1	10	High-speed peck drilling cycle (FS15-T format)
G83.5	G83.5	G83.5	10	High-speed peck drilling cycle
G83.6	G83.6	G83.6	10	Peck drilling cycle
G84	G84	G84	10	Cycle for face tapping
G84.2	G84.2	G84.2	10	Rigid tapping cycle (FS15-T format)
G85	G85	G85	10	Cycle for face boring
G87	G87	G87	10	Cycle for side drilling
G87.5	G87.5	G87.5	10	High-speed peck drilling cycle
G87.6	G87.6	G87.6	10	Peck drilling cycle
G88	G88	G88	10	Cycle for side tapping
G89	G89	G89	10	Cycle for side boring
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21	01	Threading cycle
G94	G79	G24	01	End face turning cycle
G91.1	G91.1	G91.1	00	Maximum specified incremental amount check
G96	G96	G96	02	Constant surface speed control
G96.1	G96.1	G96.1	00	Spindle indexing execution (waiting for completion)
G96.2	G96.2	G96.2	00	Spindle indexing execution (not waiting for completion)
G96.3	G96.3	G96.3	00	Spindle indexing completion check
G96.4	G96.4	G96.4	00	SV speed control mode ON
G97	G97	G97	02	Constant surface speed control cancel
G98	G94	G94		Feed per minute
G99	G95	G95		Feed per revolution
-	G90	G90	03	Absolute programming
-	G91	G91	03	Incremental programming
-	G98	G98	11	Canned cycle : return to initial level
-	G99	G99	11	Canned cycle : return to R point level

12.2.2 G Code List in the Machining Center System

M

Table 12.2.2 (a) G code list

G code	Group	Function	
G00	01	Positioning (rapid traverse)	
G01	01	Linear interpolation (cutting feed)	
G02	01	Circular interpolation CW or helical interpolation CW	
G03	01	Circular interpolation CCW or helical interpolation CCW	
G04	00	Dwell	
G05	00	AI contour control (high-precision contour control compatible command)	
G05.1	00	AI contour control / Smooth tolerance+ control	
G05.4	00	HRV3 on/off	
G07.1 (G107)	00	Cylindrical interpolation	
G08	00	AI contour control (advanced preview control compatible command)	
G09	00	Exact stop	
G10	00	Programmable data input	
G10.6	00	Tool retract and recover	
G10.8	00	Programmable internal data change	
G11	00	Programmable data input mode cancel	
G15	17	Polar coordinates command cancel	
G16	17	Polar coordinates command	
G17	02	XpYp plane selection	Xp: X axis or its parallel axis

G code	Group	Function	
G18	02	ZpXp plane selection	Yp: Y axis or its parallel axis
G19	02	YpZp plane selection	Zp: Z axis or its parallel axis
G20 (G70)	06	Input in inch	
G21 (G71)	06	Input in mm	
G22	04	Stored stroke check function on	
G23	04	Stored stroke check function off	
G25	19	Spindle speed fluctuation detection off	
G26	19	Spindle speed fluctuation detection on	
G27	00	Reference position return check	
G28	00	Automatic return to reference position	
G28.2	00	In-position check disable reference position return	
G29	00	Movement from reference position	
G30	00	2nd, 3rd and 4th reference position return	
G30.2	00	In-position check disable 2nd, 3rd, or 4th reference position return	
G31	00	Skip function	
G31.8	00	EGB-axis skip	
G33	01	Threading	
G37	00	Automatic tool length measurement	
G38	00	Tool radius/tool nose radius compensation : preserve vector	
G39	00	Tool radius/tool nose radius compensation : corner circular interpolation	
G40	07	Tool radius/tool nose radius compensation : cancel	
G41	07	Tool radius/tool nose radius compensation : left	
G42	07	Tool radius/tool nose radius compensation : right	
G40.1	18	Normal direction control cancel mode	
G41.1	18	Normal direction control on : left	
G42.1	18	Normal direction control on : right	
G43	08	Tool length compensation +	
G44	08	Tool length compensation -	
G43.7	08	Tool offset	
G45	00	Tool offset : increase	
G46	00	Tool offset : decrease	
G47	00	Tool offset : double increase	
G48	00	Tool offset : double decrease	
G49 (G49.1)	08	Tool length compensation cancel	
G50	11	Scaling cancel	
G51	11	Scaling	
G50.1	22	Programmable mirror image cancel	
G51.1	22	Programmable mirror image	
G50.4	00	Cancel synchronous control	
G50.5	00	Cancel composite control	
G50.6	00	Cancel superimposed control	
G51.4	00	Start synchronous control	
G51.5	00	Start composite control	
G51.6	00	Start superimposed control	
G52	00	Local coordinate system setting	
G53	00	Machine coordinate system setting	
G53.1	00	Tool axis direction control	
G53.2	00	Selecting a machine coordinate system with feedrate	
G53.6	00	Tool center point retention type tool axis direction control	

G code	Group	Function
G54 (G54.1)	14	Workpiece coordinate system 1 selection
G55	14	Workpiece coordinate system 2 selection
G56	14	Workpiece coordinate system 3 selection
G57	14	Workpiece coordinate system 4 selection
G58	14	Workpiece coordinate system 5 selection
G59	14	Workpiece coordinate system 6 selection
G60	00	Single direction positioning
G61	15	Exact stop mode
G62	15	Automatic corner override
G63	15	Tapping mode
G64	15	Cutting mode
G65	00	Macro call
G66	12	Macro modal call A
G66.1	12	Macro modal call B
G67	12	Macro modal call A/B cancel
G68	16	Coordinate system rotation start or 3-dimensional coordinate conversion mode on
G69	16	Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off
G68.2	16	Tilted working plane indexing
G68.3	16	Tilted working plane indexing by tool axis direction
G68.4	16	Tilted working plane indexing (incremental multi-command)
G70.7	00	Finishing cycle
G71.7	00	Outer surface rough machining cycle
G72.7	00	End rough machining cycle
G73.7	00	Closed loop cutting cycle
G74.7	00	End cutting off cycle
G75.7	00	Outer or inner cutting off cycle
G76.7	00	Multiple threading cycle
G72.1	00	Figure copying (rotary copy)
G72.2	00	Figure copying (linear copy)
G73	09	Peck drilling cycle
G74	09	Left-handed tapping cycle
G75	01	Plunge grinding cycle
G76	09	Fine boring cycle
G77	01	Plunge direct sizing/grinding cycle
G78	01	Continuous-feed surface grinding cycle
G79	01	Intermittent-feed surface grinding cycle
G80	09	Canned cycle cancel Electronic gear box : synchronization cancellation
G80.4	34	Electronic gear box: synchronization cancellation
G80.5	24	Electronic gear box 2 pair: synchronization cancellation
G81	09	Drilling cycle or spot boring cycle Electronic gear box : synchronization start
G81.4	34	Electronic gear box: synchronization start
G81.5	24	Electronic gear box 2 pair: synchronization start
G82	09	Drilling cycle or counter boring cycle
G83	09	Peck drilling cycle
G84	09	Tapping cycle
G84.2	09	Rigid tapping cycle (FS15 format)
G84.3	09	Left-handed rigid tapping cycle (FS15 format)
G85	09	Boring cycle
G86	09	Boring cycle
G87	09	Back boring cycle
G88	09	Boring cycle
G89	09	Boring cycle
G90	03	Absolute programming

G code	Group	Function
G91	03	Incremental programming
G91.1	00	Checking the maximum incremental amount specified
G92	00	Setting for workpiece coordinate system or clamp at maximum spindle speed
G92.1	00	Workpiece coordinate system preset
G93	05	Inverse time feed
G94	05	Feed per minute
G95	05	Feed per revolution
G96	13	Constant surface speed control
G97	13	Constant surface speed control cancel
G96.1	00	Spindle indexing execution (waiting for completion)
G96.2	00	Spindle indexing execution (not waiting for completion)
G96.3	00	Spindle indexing completion check
G96.4	00	SV speed control mode ON
G98	10	Canned cycle : return to initial level
G99	10	Canned cycle : return to R point level
G107	00	Cylindrical interpolation
G160	20	Infeed control cancel
G161	20	Infeed control

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3401	GSC	GSB						

[Input type] Parameter input
[Data type] Bit path

#6 GSB The G code system is set.
#7 GSC

GSC	GSB	G code
0	0	G code system A
0	1	G code system B
1	0	G code system C

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23	CLR		FPM	G91			G01
	G23	CLR	G70		G91	G19	G18	G01

[Input type] Parameter input
[Data type] Bit path

#0 G01 Mode entered when the power is turned on or when the control is cleared
0: G00 mode (positioning)
1: G01 mode (linear interpolation)

#1 G18 Plane selected when power is turned on or when the control is cleared
0: G17 mode (plane XY)
1: G18 mode (plane ZX)

#2 G19 Plane selected when power is turned on or when the control is cleared
0: The setting of bit 1 (G18) of parameter No. 3402 is followed.
1: G19 mode (plane YZ)
When this bit is set to 1, set bit 1 (G18) of parameter No. 3402 to 0.

#3 G91 When the power is turned on or when the control is cleared
0: G90 mode (absolute programming)
1: G91 mode (incremental programming)

#4 FPM At power-on time or in the cleared state:
0: G99 or G95 mode (feed per revolution) is set.
1: G98 or G94 mode (feed per minute) is set.

#5 G70 The commands for inch input and metric input are:
0: G20 (inch input) and G21 (metric input).
1: G70 (inch input) and G71 (metric input).

#6 CLR Reset button on the MDI unit, external reset signal, reset and rewind signal, and emergency stop signal
0: Cause reset state.
1: Cause clear state.
For the reset and clear states, refer to Appendix in the OPERATOR'S MANUAL.

#7 G23 When the power is turned on
0: G22 mode (stored stroke check on)
1: G23 mode (stored stroke check off)

	#7	#6	#5	#4	#3	#2	#1	#0
4902								G26
								G26

[Input type] Parameter input
[Data type] Bit path

#0 G26 When power is turned on or when the control is cleared:
0: G25 mode (spindle speed fluctuation detection is disabled) is selected.
1: G26 mode (spindle speed fluctuation detection is enabled) is selected.

Alarm and message

Number	Message	Description
PS0010	IMPROPER G-CODE	An unusable G code is specified.

Note

NOTE

1 When the power is turned on or the cleared state is set by a reset (bit 6 (CLR) of parameter No. 3402 is set to 1), modal G codes are placed in the following states:
(1) G codes marked with ◤ in G code lists are enabled.
(2) When the system is cleared due to power-on or reset, whichever specified, either G20 or G21, remains effective.
(3) Bit 7 (G23) of parameter No. 3402 is used to specify whether G22 or G23 is to be selected upon power-on. The selection of G22 or G23 is not, however, changed when the CNC is cleared upon a reset. When the system is cleared due to reset, whichever specified, either G22 or G23, remains effective.

2 G codes of group 00 except G10 and G11 are single-shot G codes.

NOTE

3 Alarm PS0010 is displayed when a G code not listed in the G code list is specified or a G code without a corresponding option is specified.
4 G codes of different groups can be specified in the same block.
If G codes of the same group are specified in the same block, the G code specified last is valid.
5 If a G code of group 01 is specified in a canned cycle for drilling, the canned cycle is canceled in the same way as when a G80 command is specified. G codes of group 01 are not affected by G codes for specifying a canned cycle.
6 When G code system A is used for a canned cycle for drilling, only the initial level is provided at the return point.
7 G codes are displayed for each group number.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Preparatory function (G function)
	SETTINGS ATPOWER-ON, IN THE CLEAR STATE, OR IN TH ERESET STATE

12.3 PROGRAM CONFIGURATION

Overview

A program consists of the following components:

Table 12.3 (a) Program components

Components	Descriptions
Program code start	Symbol indicating the start of a program file
Leader section	Used for the title of a program file, etc.
Program start	Symbol indicating the start of a program
Program section	Commands for actual machining
Comment section	Comments or directions for the operator
Program code end	Symbol indicating the end of a program file

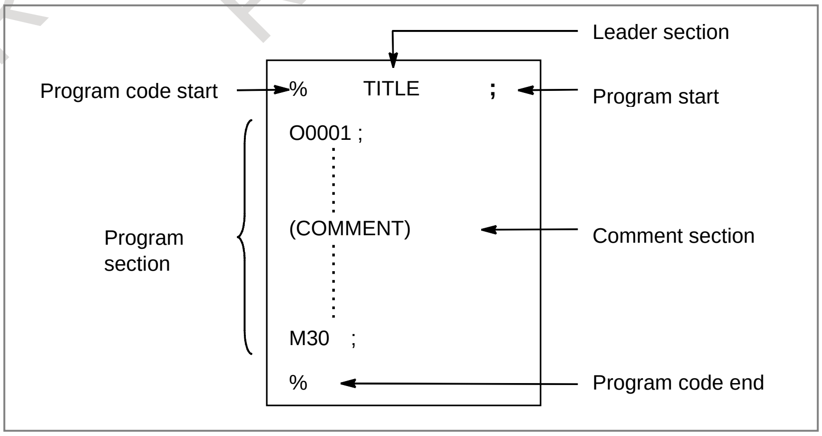


Fig. 12.3 (a) Program configuration

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0100							CTV	

[Input type] Setting input
[Data type] Bit

#1 CTV Character counting for TV check in the comment section of a program.
0: Performed
1: Not performed

	#7	#6	#5	#4	#3	#2	#1	#0
3201		NPE	N99					

[Input type] Parameter input
[Data type] Bit path

#5 N99 With an M99 block, when bit 6 (NPE) of parameter No. 3201 is set to 0, program registration is assumed to be:
0: Completed
1: Not completed

#6 NPE With an M02, M30, or M99 block, program registration is assumed to be:
0: Completed
1: Not completed

6030	M code to execute external device subprogram calls

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 99999999

Set the M code to execute external device subprogram calls. When 0 is set, M198 is used. M01, M02, M30, M98, and M99 cannot be used to execute external device subprogram calls. When a negative number, 1, 2, 30, 98, or 99 is set for this parameter, M198 is used to execute external device subprogram calls.

Alarm and message

Number	Message	Description
SR0001	TH ERROR	A TH error was detected during reading from an input device. The read code that caused the TH error and how many statements it is from the block can be verified in the diagnostics screen.
SR0002	TV ERROR	An error was detected during the single–block TV error. The TV check can be suppressed by setting bit 0 (TVC) of parameter No. 0000 to 0.
PS5010	END OF RECORD	The EOR (End of Record) code is specified in the middle of a block. This alarm is also generated when the percentage at the end of the NC program is read.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	PROGRAM CONFIGURATION

12.4 PART PROGRAM STORAGE SIZE / NUMBER OF REGISTERABLE PROGRAMS

The following table lists the combinations of program storage sizes and the total number of registerable programs.

Part program storage size	Number of registerable programs expansion 1
2Mbyte	1000

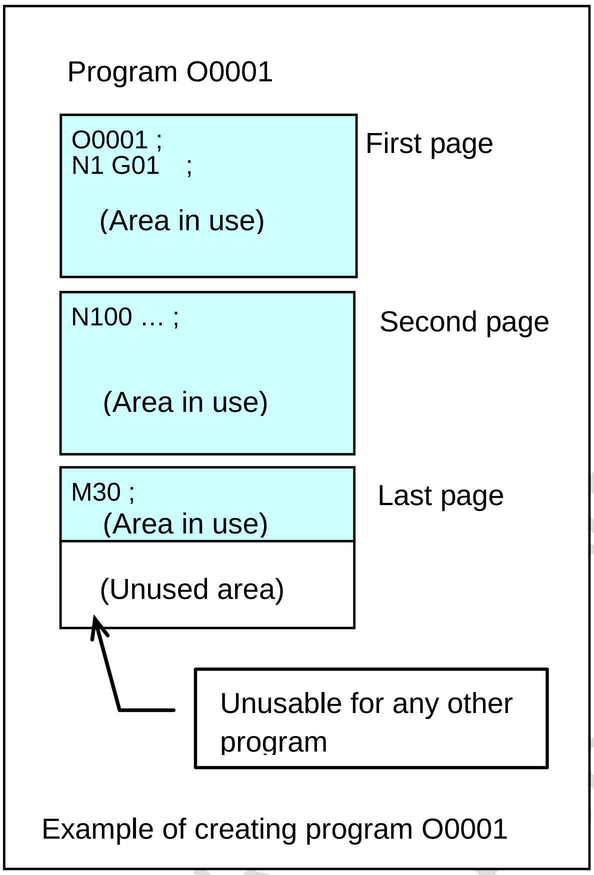


Fig. 13.4

> **NOTE**
> 1 Creating one folder results in the number of programs yet to be registerable decreasing one.
> 2 The program storage size means the maximum size of a program if the program is the one and only program registered.
> 3 If more than one program is registered, the total size of registerable programs reduces for the following reason.
> The FS0*i*-F Plus manage programs in page units. The unit of program storage is managed also in page units. When a program is created, as many pages as necessary to store the program are secured, and the program is stored on these pages. Generally, the last program storage page has an unused area (Fig. 13.4). This unused area cannot be used to store any other program. For the sake of program management, it is regarded as an area in use.
> The FS0*i*- C uses a similar way of management, but the unit of pages in it differs from that in the FS0*i*-F Plus. So, if more than one program is registered in the FS0*i*-F Plus, the total program size of registerable programs in the FS0*i*-F Plus differs from that in the FS0*i*-C.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Program management

12.5 INCH/METRIC CONVERSION

Overview

Either inch or metric input (least input increment) can be switched by the following five methods.

- G code(G20/G21)
- SETTING (HANDY) screen/setting bit 2 (INI) parameter No. 0000
- PMC Window
- Programable parameter input(G10)
- Custom macro variable #3005

Explanation

- Switching method using a G code(G20/G21)

The unit of input data is switched by G20/G21. It switches to inch input by G20, and metric input by G21. Please refer to the OPERATOR'S MANUAL (Common to Lathe System/Machining Center System)(B-64694EN) for switching conditions of G20/G21.

- Switching method using a SETTING (HANDY) screen/setting parameter

The unit of input data is switched by SETTING (HANDY) screen or setting bit 2 (INI) parameter No. 0000. Please refer to the OPERATOR'S MANUAL (Common to Lathe System/Machining Center System)(B-64694EN) for switching conditions.

- Switching method using a PMC Window

The unit of input data is switched by writing setting data of PMC window (low speed type). Please refer to the PMC Programming Manual (B-64513EN) for details.

- Switching method using a programable parameter input(G10)

Setting bit 2 (INI) parameter No. 0000 is rewritten by the G10 command, and the unit of input data is switched.

- Switching method using a custom macro variable

The unit of input data is switched by system variable #3005.

- Switching position

The unit of input data is switched at the reference position of the machine coordinate system origin. When the bit 1 (INAx) of parameter No. 14000 set to 1, if an inch-metric switch command is executed at a position other than the machine coordinate system origin (position = zero), the alarm PS5362, “CONVERT INCH/MM AT REF-POS” is generated.

- In case G code(G20/G21), it is possible to change the unit at the positions other than 0 of the machine coordinate by the following settings.

(a) Setting bit 2 (IRFx) of parameter No. 14000 to 1 enables inch/metric conversion to be performed in the reference position (parameter No. 1240). However, enable a function of workpiece coordinate system (bit 0 (NWZ) of parameter No. 8136 is 0).

(b) Setting bit 0 (NIM) of parameter No. 11222 enables inch/metric conversion to be performed even in positions other than the reference position. However, enable functions of workpiece coordinate system and workpiece coordinate system preset (bit 0 (NWZ) of parameter No. 8136 is 0 and bit 1 (NWC) of parameter No. 8136 is 0).

- In case SETTING (HANDY) screen or setting parameter or PMC Window, it is possible to change the unit at the positions other than 0 of the machine coordinate by the following settings.

(c) Setting bit 0 (NIM) of parameter No. 11222 enables inch/metric conversion to be performed even in positions other than the reference position. However, enable functions of workpiece coordinate system and workpiece coordinate system preset (bit 0 (NWZ) of parameter No. 8136 is 0 and bit 1 (NWC) of parameter No. 8136 is 0).

(d) Setting bit 0 (NIM) of parameter No. 11222 and bit 3 (IMRx) of parameter No. 14000 enables inch/metric conversion to limit in the reference position (parameter No. 1240). However, enable functions of workpiece coordinate system and workpiece coordinate system preset (bit 0 (NWZ) of parameter No. 8136 is 0 and bit 1 (NWC) of parameter No. 8136 is 0).

Table 12.5 (a) The position that inch / metric conversion can be changed by G code(G20/G21)

Parameter No.				Position that can be changed		
14000			11222	Reference point (parameter No. 1240)		Other than the reference position
#1(INAx)	#2(IRFx)	#3(IMRx)	#0(NIM)	=0	≠0	
0	0	0	0	Changed	Not changed	Not changed
1	0	0	0	Changed	Not changed (*1)	Not changed (*1)
-	1	0	0	Changed	Changed	Not changed (*1)
-	-	0	1	Changed	Changed	Changed
-	-	1	1	Changed	Changed	Not changed (*1)

(*1) Alarm(PS5362)

Table 12.5 (b) The position that inch / metric conversion can be changed by SETTING (HANDY) screen or setting parameter or PMC Window

Parameter No.				Position that can be changed		
14000			11222	Reference point (parameter No. 1240)		Other than the reference position
#1(INAx)	#2(IRFx)	#3(IMRx)	#0(NIM)	=0	≠0	
0	0	0	0	Changed	Not changed	Not changed
1	0	0	0	Changed	Not changed (*1)	Not changed (*1)
-	1	0	0	Changed	Not changed	Not changed (*1)
-	-	0	1	Changed	Changed	Changed
-	-	1	1	Changed	Changed	Not changed (*1)

(*1) Alarm(PS5362)

Signal

Inch input signal INCH<Fn002.0>

[Classification] Output signal

[Function] This signal informs that inch input mode is active.

[Output cond.] 1indicates that the inch input mode (G20) is in progress, and 0 indicates that metric input mode (G21) is in progress.

> **NOTE**
> This signal changes to the corresponding state when the following methods.
> - SETTING (HANDY) screen/setting bit 2 (INI) parameter No. 0000.
> - Parameter INI is rewritten by Programable parameter input (G10)
> - System variable #3005

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn002								INCH

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0000						INI		

[Input type] Setting input
[Data type] Bit path

#2 INI Unit of input
0: In metrics
1: In inches

	#7	#6	#5	#4	#3	#2	#1	#0
1001								INM

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 INM Least command increment on the linear axis
0: In mm (metric system machine)
1: In inches (inch system machine)

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx
#1 ROSx Setting linear or rotary axis.

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)

ROSx	ROTx	Meaning
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values and relative coordinate values can be selected without rounding / rounding according to bits 0 (ROAx) and 2 (RRLx) of parameter No. 1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

1240	Coordinate value of the reference position in the machine coordinate system

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the reference position in the machine coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
3104								MCN

[Input type] Parameter input
[Data type] Bit path

#0 MCN Machine position
0: Regardless of whether input is made in mm or inches, the machine position is displayed in mm for millimeter machines, or in inches for inch machines.
1: When input is made in mm, the machine position is displayed in mm, and when input is made in inches, the machine position is displayed in inches accordingly.

	#7	#6	#5	#4	#3	#2	#1	#0
3405								AUX

[Input type] Parameter input
[Data type] Bit path

#0 AUX When the second auxiliary function is specified in the calculator-type decimal point input format or with a decimal point, the multiplication factor for a value output (onto the code signal) relative to a specified value is such that:
0: The same multiplication factor is used for both of metric input and inch input.
1: A multiplication factor used for inch input is 10 times greater than that used for metric input.

When the second auxiliary function is specified in the calculator-type decimal point input format or with a decimal point, the value output onto the code signal is a specified value multiplied by a value indicated below.

Increment system		Parameter AUX=0	Parameter AUX=1
Metric input system	IS-A for reference axis	100 times	100 times
	IS-B for reference axis	1000 times	1000 times
	IS-C for reference axis	10000 times	10000 times
Inch input system	IS-A for reference axis	100 times	1000 times
	IS-B for reference axis	1000 times	10000 times
	IS-C for reference axis	10000 times	100000 times

	#7	#6	#5	#4	#3	#2	#1	#0
11222						IMG	CIM	NIM

[Input type] Parameter input
[Data type] Bit path

#0 NIM Automatic conversion of a coordinate system by an inch/metric conversion command (G20 or G21) is:
0: Not performed.
1: Performed.

#1 CIM When an inch/metric conversion command (G20 or G21) is specified, if the workpiece coordinate system is shifted by the shift amount as described below:
0: An alarm PS1298”ILLEGAL INCH/METRIC CONVERSION” is issued.
1: Clearing of the shift amount is performed.

If bit 0 (NIM) of parameter No. 11222 is set to 1, or if bit 2 (IRF) of parameter No. 14000 is set to 1, this parameter clears the following:
- Manual intervention made when the manual absolute signal*ABSM<Gn006.2> is off
- Issuance of a move command with the machine locked
- Movement by handle interrupt
- Operation with a mirror image
- Shifting of a workpiece coordinate system when a local coordinate system or workpiece coordinate system is set up

#2 IMG Inch/metric conversion is:
0: Performed with the G20/G21 (G70/G71).
1: Not performed with the G20/G21 (G70/G71).

> **NOTE**
> If bit 2 of parameter No. 11222 is 1 (inch/metric conversion with G20/G21 is disabled), only bit 2 of parameter No. 0 can be used to perform inch/metric conversion. If bit 2 of parameter No.0 = 0, the metric system is used. If bit 2 of parameter No.0 = 1, the inch system is used.

	#7	#6	#5	#4	#3	#2	#1	#0
14000					IMRx	IRFx	INAx	

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 INAx If an inch-metric switch command is executed at a position of the machine coordinate system origin other than zero,
0: No alarm is issued.
1: Alarm PS5362, "CONVERT INCH/MM AT REF-POS" is issued.

#2 IRFx An inch-metric switch command (G20, G21) at the reference position is:
0: Disabled.
1: Enabled.

> **NOTE**
> 1 When this function is enabled for an axis, if an attempt to switch between the inch and metric unit is made although the tool is not at the reference position on that axis, an alarm PS5362 is issued, and switching between the inch and metric unit is canceled. Be sure to move the tool to the reference position by, for example, specifying G28 before switching between the inch and metric unit.
> 2 This function enables the inch/metric switching commands (G20 and G21) at the reference position. It does not enable the switching of the setting input unit (bit 2 (INI) of parameter No.0000).
> 3 Switching between inch and metric by setting the setting input unit (bit 2 (INI) of parameter No. 0000) is enabled only when the machine coordinate of the first reference position is 0 (parameter No. 1240 is 0) and presence on the first reference position is assumed. For a system in which the machine coordinate of the first reference position is not 0, set this parameter to 1 and specify G20/G21 in the first reference position to switch between inch and metric.
> 4 When this function is disabled, the switching is enabled only when the machine coordinate of the first reference position is 0 (parameter No. 1240 is 0) and presence on the first reference position is assumed.

#3 IMRx When bit 0 (NIM) of parameter No.11222 is set to 1, the reference position return before commanding inch/metric conversion :
0: Not required.
1: Required.

NOTE
1 Check the reference position return end signals ZP1 to ZP8 <Fn094> of the axes actually moving during Composite control.
2 When parking the master axis during synchronization control, check the reference position return end signals ZP1 to ZP8<Fn094> of the slave axis.
3 Bit 3 (IMRx) of parameter No.14000 is not set for the synchronous control on the slave axis. When IMRx = 1, the reference position return end signals ZP1 to ZP8<Fn094> of the slave axis must be "1".

Warning

⚠ **WARNING**
When switching inch input (G20) to metric input (G21) and vice versa, tool compensation values are automatically converted.

Note

NOTE
1 When the least input increment and the least command increment systems are different, the maximum error is half of the least command increment. This error is not accumulated.
(Example)
- Increment system : IS-B
- Least input increment (for input uit) : In case of "0.0001" inch
- Least command increment (for machine unit) : "0.00254" -> "0.003"
(Difference =
0.00046 : About 1/2 of least command increment)
Input "0.0001 inch" -> the moving increment of the machine "0.00254 mm".
However, when the increment system is IS-B, the value of the movement of the machine becomes "0.003 mm".
Therefore, the error margin of "0.00046 mm" (About 1/2 of least command increment) is generated as a difference.
2 The inch and metric input can also be switched using bit 2 (INI) of parameter No. 0000.
3 If a function selected using bit 2 (IRF) of parameter No. 14000 or bit 0 (NIM) of parameter No. 11222 is not used, be sure to perform inch/metric switching at the reference position (machine coordinate system origin).

Alarm and message

Number	Message	Description
PS1298	ILLEGAL INCH/METRIC CONVERSION	(1) When the bit 1 (CIM) of parameter No. 11222 set to 0, and the workpiece coordinate system has been shifted from the machine coordinate system by the following, the inch/metric conversion has been performed. - Manual intervention performed with the manual absolute signal being off - Machine locked - Handle interrupt - Mirror image - Workpiece coordinate system shift caused by local coordinate system setting (G52) or workpiece coordinate system setting (G92) (2) Inch/metric conversion was executed in a modal state where it can not command. Check the modal state. (3) Inch/metric conversion can not be commanded with programmable parameter input or # 3005 (setting data). It commands G20/G21 or G70/G71. (4) During automatic operation, inch / metric conversion by MDI input or PMC window was done. Stop automatic operation and command.
PS5362	CONVERT INCH/MM AT REF-POS	(1) When the bit 1 (INAx) of parameter No. 14000 set to 1, the inch-metric switch command is executed at a position of the machine coordinate system origin other than zero. (2) When the bit 3 (IMRx) or bit 2 (IRFx) of parameter No. 14000 set to 1, the inch-metric switch command is executed at the other than reference position (parameter No. 1240). Perform an inch/metric conversion after returning to the reference position. (3) When bit 0 (NIM) of parameter No.11222 set to 1, the inch/metric conversion by programmable parameter input or #3005 (setting data) was executed at a position other than the machine coordinate system origin (position = zero). Please command it at the machine coordinate zero position.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Inch metric conversion (G20, G21)

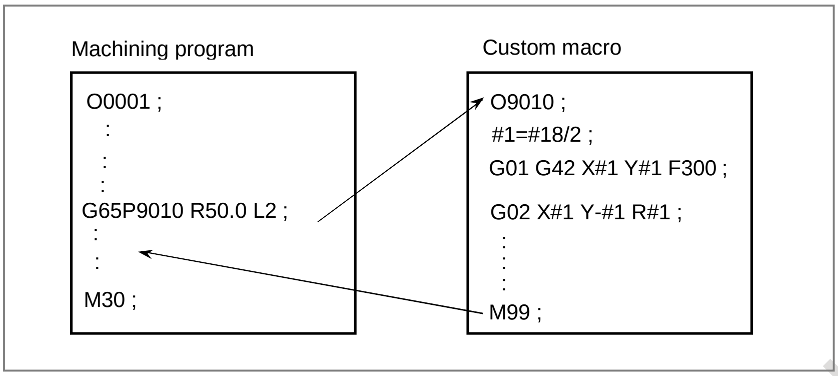
12.6 CUSTOM MACRO

12.6.1 Custom Macro

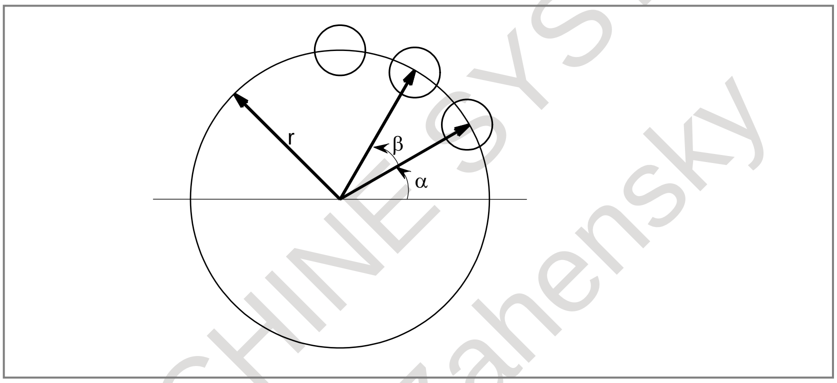
Overview

A subprogram is useful for repeatedly executing the same operation. With the custom macro function, however, variables, operation commands, conditional branches, and so forth can be used to create general-purpose programs more simply for pocketing, user-specific canned cycles, and so forth.

From a machining program, a custom macro can be called with a simple instruction as in the case of a subprogram.



Thus, when a function is programmed as a custom macro body, the function can be made more universal. This means that a program can be created using variables for values that can change or are not defined yet. This feature also leads to application of group technology.
For example, classify similar workpieces into a group and create a general-purpose custom macro by using variables for such a group. Thus, the need for programming for an individual workpiece in the group can be eliminated by assigning desired values to the variables.



As an example, the bolt hole circle shown above can be created simply. By creating and once registering a custom macro for the bolt hole circle, the CNC can operate as if it had a bolt hole circle function.
The programmer can use the bolt hole circle function by remembering the following command:

G65 Pp Rr Aα Bβ Kk ;
P : Macro number of bolt hole circle
r : Radius
α: Start angle
β: Angle between circles
k : Number of circles

Signal

Custom Macro Input Signals

UI000 to UI015<Gn054,Gn055>, UI016 to UI031<Gn056,Gn057>, UI100 to UI131<Gn276 to Gn279>, UI200 to UI231<Gn280 to Gn283> UI300 to UI331<Gn284 to Gn287>

[Classification] Input signal

[Function] No function is provided for the control unit. These signals can be read by a custom macro as a type of system variable, and are used for interface between custom macros and the PMC.
These signals correspond to system variables as indicated below.

- When the bit 0 (MIF) of parameter No. 6001 is set to 0 :

Signals	Q'ty	Variables	Correspondence of values
UI000	1	#1000	
UI001	1	#1001	
UI002	1	#1002	
UI003	1	#1003	0 at 0 and 1 at 1
:	:	:	
UI014	1	#1014	
UI015	1	#1015	
UI000 to UI015	16	#1032	Unsigned 16-bit binary code *1

*1 Variable value #1032 = $\sum_{i=0}^{15}\{\#[1000+i]\times 2^{i}\}$

These system variables cannot be used on the left side of an assignment statement.

- When the bit 0 (MIF) of parameter No. 6001 is set to 1 :

Signals	Q'ty	Variables	Correspondence of values
UI000	1	#1000	
UI001	1	#1001	
UI002	1	#1002	
UI003	1	#1003	0 at 0 and 1 at 1
:	:	:	
UI030	1	#1030	
UI031	1	#1031	
UI000 to UI031	32	#1032	Signed 32-bit binary code *1
UI100 to UI131	32	#1033	Signed 32-bit binary code *1
UI200 to UI231	32	#1034	Signed 32-bit binary code *1
UI300 to UI331	32	#1035	Signed 32-bit binary code *1

*1 Variable value #(1032+n) = $\sum_{i=0}^{30}\{2^{i}\times V_{i}\}-2^{31}\times V_{31}$

Where Vi=0 when UIni is 0 and Vi=1 when UIni is 1

n : 0 to 3

These system variables cannot be used on the left side of an assignment statement.

Custom Macro Output Signals

UO000 to UO015<Fn054,Fn055>, UO100 to UO131<Fn056 to Fn059>
UO016 to UO031<Fn276,Fn277>, UO200 to UO231<Fn280 to Fn283>
UO300 to UO331<Fn284 to Fn287>

[Classification] Output signal

[Function] No function is provided for the control unit. These signals can be read or written by a custom macro as a type of system variable, and are used for interface between custom macros and the PMC.

These signals correspond to system variables as indicated below.

- When the bit 0 (MIF) of parameter No. 6001 is set to 0 :

Signals	Q'ty	Variables	Correspondence of values
UO000	1	#1100	
UO001	1	#1101	
UO002	1	#1102	
UO003	1	#1103	0 at 0 and 1 at 1
:	:	:	
UO014	1	#1114	
UO015	1	#1115	
UO000 to UO015	16	#1132	Unsigned 16-bit binary code *1
UO100 to UO131	32	#1133	Signed 32-bit binary code *2

*1 Variable value #1132 = $\sum_{i=0}^{15}\left\{\#[1100+i]\times 2^{i}\right\}$

*2 Variable value #1133 = $\sum_{i=0}^{30}\left\{2^{i}\times V_{i}\right\}-2^{31}\times V_{31}$

Where Vi=0 when UO1i is 0and Vi=1 when UO1i is 1
These system variables can be used on the left side of an assignment statement as well as on the right side.
The value assigned to the system variable used on the left side last is used for the value of the system variable to be assigned (sent) on the right side.

- When the bit 0 (MIF) of parameter No. 6001 is set to 1 :

Signals	Q'ty	Variables	Correspondence of values
UO000	1	#1100	0 at 0 and 1 at 1
UO001	1	#1101	
UO002	1	#1102	
UO003	1	#1103	
:	:	:	
UO030	1	#1130	
UO031	1	#1131	
UO000 to UO031	32	#1132	Signed 32-bit binary code *1
UO100 to UO131	32	#1133	Signed 32-bit binary code *1
UO200 to UO231	32	#1134	Signed 32-bit binary code *1
UO300 to UO331	32	#1135	Signed 32-bit binary code *1

*1 Variable value #(1132+n) = $\sum_{i=0}^{30}\left\{2^{i}\times V_{i}\right\}-2^{31}\times V_{31}$

Where Vi=0 when UOni is 0and Vi=1 when UOni is 1
n : 0 to 3

These system variables can be used on the left side of an assignment statement as well as on the right side.
The value assigned to the system variable used on the left side last is used for the value of the system variable to be assigned (sent) on the right side.

Custom Macro Input Signals for Interface Signal R Addresses
UI400 to UI431 <Rr to Rr+3>
UI500 to UI531 <Rr+4 to Rr+7>
UI600 to UI631 <Rr+8 to Rr+11>
UI700 to UI731 <Rr+12 to Rr+15>

[Classification] Input signal

[Function] No function is provided for the control unit. These signals can be read by a custom macro as a type of system variable, and are used for interface between custom macros and the PMC.
These signals correspond to system variables as indicated below.

Signals	Q'ty	Variables	Correspondence of values
UI400	1	#1036	0 at 0 and 1 at 1
UI401	1	#1037	
:	:	:	
UI431	1	#1067	
UI400 to UI431	32	#1068	Signed 32-bit binary code
UI500 to UI531	32	#1069	Signed 32-bit binary code
UI600 to UI631	32	#1070	Signed 32-bit binary code
UI700 to UI731	32	#1071	Signed 32-bit binary code

The letter r represents a setting of parameter No. 6093.
These system variables cannot be used on the left side of an assignment statement.

> **NOTE**
> These signals are available when bit 2 (IFR) of parameter No. 6020 is set to 1.

> ⚠ **WARNING**
> To set bit 2 (IFR) of parameter No. 6020 to 1, set an appropriate value to parameter No.6094 beforehand.
> When parameter No.6094 is 0, R0~ of an internal relay is set. If the internal relay R0~ is used to another area, the operation not intended might be done.

Custom Macro Output Signals for Interface Signal R Addresses
UO400 to UO431 <Rr to Rr+3>
UO500 to UO531 <Rr+4 to Rr+7>
UO600 to UO631 <Rr+8 to Rr+11>
UO700 to UO731 <Rr+12 to Rr+15>

[Classification] Output signal

[Function] No function is provided for the control unit. These signals can be read or written by a custom macro as a type of system variable, and are used for interface between custom macros and the PMC.
These signals correspond to system variables as indicated below.

Signals	Q'ty	Variables	Correspondence of values
UO400 UO401 : UO431	1 1 : 1	#1136 #1137 : #1167	0 at 0 and 1 at 1
UO400 to UO431	32	#1168	Signed 32-bit binary code
UO500 to UO531	32	#1169	Signed 32-bit binary code
UO600 to UO631	32	#1170	Signed 32-bit binary code
UO700 to UO731	32	#1171	Signed 32-bit binary code

The letter r represents a setting of parameter No. 6094.
These system variables can be used on the left side of an assignment statement as well as on the right side. Using these system variables on the right side enables the last value assigned (sent) to each variable on the left side to be read.

> **NOTE**
> These signals are available when bit 2 (IFR) of parameter No. 6020 is set to 1.

> ⚠ **WARNING**
> To set bit 2 (IFR) of parameter No. 6020 to 1, set an appropriate value to parameter No.6094 beforehand.
> When parameter No.6094 is 0, R0~ of an internal relay is set. If the internal relay R0~ is used to another area, the operation not intended might be done.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn054	UI007	UI006	UI005	UI004	UI003	UI002	UI001	UI000
Gn055	UI015	UI014	UI013	UI012	UI011	UI010	UI009	UI008
Gn056	UI023	UI022	UI021	UI020	UI019	UI018	UI017	UI016
Gn057	UI031	UI030	UI029	UI028	UI027	UI026	UI025	UI024
Gn276	UI107	UI106	UI105	UI104	UI103	UI102	UI101	UI100
Gn277	UI115	UI114	UI113	UI112	UI111	UI110	UI109	UI108
Gn278	UI123	UI122	UI121	UI120	UI119	UI118	UI117	UI116
Gn279	UI131	UI130	UI129	UI128	UI127	UI126	UI125	UI124
Gn280	UI207	UI206	UI205	UI204	UI203	UI202	UI201	UI200
Gn281	UI215	UI214	UI213	UI212	UI211	UI210	UI209	UI208
Gn282	UI223	UI222	UI221	UI220	UI219	UI218	UI217	UI216
Gn283	UI231	UI230	UI229	UI228	UI227	UI226	UI225	UI224
Gn284	UI307	UI306	UI305	UI304	UI303	UI302	UI301	UI300
Gn285	UI315	UI314	UI313	UI312	UI311	UI310	UI309	UI308
Gn286	UI323	UI322	UI321	UI320	UI319	UI318	UI317	UI316
Gn287	UI331	UI330	UI329	UI328	UI327	UI326	UI325	UI324

	#7	#6	#5	#4	#3	#2	#1	#0
Fn054	UO007	UO006	UO005	UO004	UO003	UO002	UO001	UO000
Fn055	UO015	UO014	UO013	UO012	UO011	UO010	UO009	UO008
Fn056	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
Fn057	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
Fn058	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
Fn059	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124
Fn276	UO023	UO022	UO021	UO020	UO019	UO018	UO017	UO016
Fn277	UO031	UO030	UO029	UO028	UO027	UO026	UO025	UO024
Fn280	UO207	UO206	UO205	UO204	UO203	UO202	UO201	UO200

Fn281	UO215	UO214	UO213	UO212	UO211	UO210	UO209	UO208
Fn282	UO223	UO222	UO221	UO220	UO219	UO218	UO217	UO216
Fn283	UO231	UO230	UO229	UO228	UO227	UO226	UO225	UO224
Fn284	UO307	UO306	UO305	UO304	UO303	UO302	UO301	UO300
Fn285	UO315	UO314	UO313	UO312	UO311	UO310	UO309	UO308
Fn286	UO323	UO322	UO321	UO320	UO319	UO318	UO317	UO316
Fn287	UO331	UO330	UO329	UO328	UO327	UO326	UO325	UO324

(The letter r represents a setting of parameter No. 6093.)

	#7	#6	#5	#4	#3	#2	#1	#0
Rr	UI407	UI406	UI405	UI404	UI403	UI402	UI401	UI400
Rr+1	UI415	UI414	UI413	UI412	UI411	UI410	UI409	UI408
Rr+2	UI423	UI422	UI421	UI420	UI419	UI418	UI417	UI416
Rr+3	UI431	UI430	UI429	UI428	UI427	UI426	UI425	UI424
to								
Rr+15	UI731	UI730	UI729	UI728	UI727	UI726	UI725	UI724

(The letter r represents a setting of parameter No. 6094.)

	#7	#6	#5	#4	#3	#2	#1	#0
Rr	UO407	UO406	UO405	UO404	UO403	UO402	UO401	UO400
Rr+1	UO415	UO414	UO413	UO412	UO411	UO410	UO409	UO408
Rr+2	UO423	UO422	UO421	UO420	UO419	UO418	UO417	UO416
Rr+3	UO431	UO430	UO429	UO428	UO427	UO426	UO425	UO424
to								
Rr+15	UO731	UO730	UO729	UO728	UO727	UO726	UO725	UO724

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6000	SBV		SBM	HGO		HMC	MGO	G67
	SBV		SBM	HGO	V15	HMC	MGO	G67

[Input type] Parameter input
[Data type] Bit path

#0 G67 If the macro modal call cancel command (G67) is specified when the macro modal call mode (G66/G66.1) is not set:
0: Alarm PS1100, "CANCEL WITHOUT MODAL CALL" is issued.
1: The specification of G67 is ignored.

#1 MGO When a GOTO statement for specifying custom macro control is executed, from the start of the program to 20 sequence numbers of executed block:
0: A high-speed branch is not executed.
1: A high-speed branch is executed.

#2 HMC A custom macro is executed:
0: At a normal speed.
1: At a high-speed.

> **NOTE**
> This parameter is Automatic setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

#3 V15 As system variable numbers for tool offset:
0 : The standard system variable numbers for the Series 16 are used.
1 : The same system variable numbers as those used for the Series 15 are used.
The tables indicate the system variables for tool offset numbers 1 to 999. The values for tool offset numbers 1 to 200 can be read from or assigned to the system variables in parentheses.

		System variable number	
		V15 = 0	**V15 = 1**
Tool length offset	Geometry offset value	#11001 to #11999 (#2201 to #2400)	#10001 to #10999 (#2001 to #2200)
	Wear offset value	#10001 to #10999 (#2001 to #2200)	#11001 to #11999 (#2201 to #2400)
Tool radius offset	Geometry offset value	#13001 to #13999	#12001 to #12999
	Wear offset value	#12001 to #12999	#13001 to #13999

#4 HGO When a GOTO statement in a custom macro control command is executed, a high-speed branch to the 30 sequence numbers immediately before the executed statement is:
0: Not made.
1: Made.

#5 SBM Custom macro statement
0: Not stop the single block
1: Stops the single block
If you want to disable the single blocks in custom macro statements using system variable #3003, set this parameter to 0. If this parameter is set to 1, the single blocks in custom macro statements cannot be disabled using system variable #3003. To control single blocks in custom macro statements using system variable #3003, use bit 7 (SBV) of parameter No. 6000.

#7 SBV Custom macro statement
0: Not stop the single block
1: By system variable # 3003, single block stop control is enabled / disabled

		Bit 5 (SBM) of parameter No. 6000	
		0	**1**
Bit 7 (SBV) of parameter No. 6000	**0**	Disables single block stop.	Enables single block stop. (With variable #3003, single block stop cannot be enabled/disabled. Single block stop is enabled at all times.)
	1	Enables single block stop. (With variable #3003, single block stop can be enabled/disabled.)	

	#7	#6	#5	#4	#3	#2	#1	#0
6001		CCV	TCS	CRO	PV5		PRT	MIF

[Input type] Parameter input
[Data type] Bit path

#0 MIF The custom macro interface signals are based on:
0: Standard specification.
(The signals UI000 to UI015, UO000 to UO015, and UO100 to UO131 are used.)
1: Extended specification.
(The signals UI000 to UI031, UI100 to UI131, UI200 to UI231, UI300 to UI331, UO000 to UO031, UO100 to UO131, UO200 to UO231, and UO300 to UO331 are used.)

#1 PRT Reading zero when data is output using a DPRINT command
0: Outputs a space
1: Outputs no data

#3 PV5 Custom macro common variables:
0: #500 to #549 are output. (Note)
1: #100 to #149 and #500 to 549 are output. (Note)

> **NOTE**
> Output variables are as the table according to the combination of added options.

When the parameter PV5=0

		Addition of custom macro common variables: Disable (bit 6 (NCV) of parameter No.8135 is 1)	Addition of custom macro common variables: Enable (bit 6 (NCV) of parameter No.8135 is 0)
Option "Embedded macro"	Non	#500 to #549	#500 to #999
	Yes	#500 to #549	#500 to #999

When the parameter PV5=1

		Addition of custom macro common variables: Disable (bit 6 (NCV) of parameter No.8135 is 1)	Addition of custom macro common variables: Enable (bit 6 (NCV) of parameter No.8135 is 0)
Option "Embedded macro"	Non	#100 to #149 and #500 to #549	#100 to #199 and #500 to #999
	Yes	#100 to #149, #200 to #499 and #500 to #549	#100 to #199, #200 to #499 and #500 to #999

#4 CRO ISO code in BPRWT or DPRNT command
0: Outputs only "LF" after data is output
1: Outputs "LF" and "CR" after data is output

#5 TCS Custom macro (subprogram)
0: Not called using a T code
1: Called using a T code

#6 CCV Common variables #100 to #149(NOTE) cleared by power-off are:
0: Cleared to <null> by reset
1: Not cleared by reset

> **NOTE**
> Cleared variables are as the table according to the combination of added options.

		Addition of custom macro common variables	
		Disable (bit 6 (NCV) of parameter No.8135 is 1)	Enable (bit 6 (NCV) of parameter No.8135 is 0)
Option "Embedded macro"	Non	#100 to #149	#100 to #199
	Yes	#100 to #149 and #200 to #499	#100 to #199 and #200 to #499

	#7	#6	#5	#4	#3	#2	#1	#0
6004						VHD		NAT
			D15					NAT

[Input type] Parameter input
[Data type] Bit path

#0 NAT The results of the custom macro functions ATAN (with 2 arguments) and ASIN are specified as follows:
0: The result of ATAN is 0 to 360.0.
The result of ASIN is 270.0 to 0 to 90.0.
1: The result of ATAN is -180.0 to 0 to 180.0.
The result of ASIN is -90.0 to 0 to 90.0.

#2 VHD With system variables #5121 to #5140:
0: The tool offset value (geometry offset value) in the block currently being executed is read. (This parameter is valid only when tool geometry/tool wear compensation memories are available.)
1: An interrupt travel distance based on manual handle interrupt is read.

#5 D15 When tool compensation memory C is used, for reading or writing tool offset values (for up to offset number 200) for D code (tool radius), the same system variables, #2401 through #2800, as Series 15 are:
0: Not used.
1: Used.

When bit 3 (V15) of parameter No. 6000 is set to 1

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2401	[#_OFSDG[1]]	#2601	[#_OFSDW[1]]
2	#2402	[#_OFSDG[2]]	#2602	[#_OFSDW[2]]
3	#2403	[#_OFSDG[3]]	#2603	[#_OFSDW[3]]
:	:	:	:	:
199	#2599	[#_OFSDG[199]]	#2799	[#_OFSDW[199]]
200	#2600	[#_OFSDG[200]]	#2800	[#_OFSDW[200]]

	#7	#6	#5	#4	#3	#2	#1	#0
6005				AXM				
				AXM				

[Input type] Parameter input
[Data type] Bit path

#4 AXM Bit 4 (MSV) of parameter No.6019 and bit 4 (TSV) of parameter No.6021 are :
0: valid only during coordinate system shift type compensation.
1: valid both during coordinate system shift type compensation and axis movement type compensation.

	#7	#6	#5	#4	#3	#2	#1	#0
6007				CVA	MGE	BCS	SCS	DPG

[Input type] Parameter input
[Data type] Bit path

#0 DPG Specifies whether to allow G codes with a decimal point to be called.
0: Do not allow.
1: Allow.

#1 SCS Specifies whether to call subprograms with S codes.
0: Do not call with S codes.
1: Call with S codes.

#2 BCS Specifies whether to call subprograms with the second auxiliary function codes.
0: Do not call with the second auxiliary function codes.
1: Call with the second auxiliary function codes.

#3 MGE Specifies whether a G code modal call is made.
0: Make a call for each block (equivalent to G66.1).
1: Make a call after movement (equivalent to G66).

#4 CVA The format for macro call arguments is specified as follows:
0: Arguments are passed in NC format without modifications.
1: Arguments are converted to macro format then passed.
[Example]
When "G65 P_ X10 ;" is specified, the value in local variable #24 in the calling program is set as follows:

Command	CVA=0	CVA=1
#24	0.01	0.01
ADP[#24]	10.0	0.01

> **NOTE**
> External operations are the same unless the ADP function is used.

	#7	#6	#5	#4	#3	#2	#1	#0
6008	IJK	GMP	ADD	ISO	KOP		MCA	F16

[Input type] Parameter input
[Data type] Bit path

#0 F16 The precision of operation is based on:
0: New specification.
1: FS16*i* compatible specification.

#1 MCA A macro alarm specification based on system variable #3000 is selected as follows:
0: An alarm number obtained by adding 3000 to a value assigned to variable #3000 and the corresponding message are displayed. (A value from 0 to 200 can be assigned to variable #3000.)
1: A value assigned to variable #3000 and the corresponding message are displayed. (A value from 0 to 4095 can be assigned to variable #3000.)

[Example]
Execution of #3000=1 (ALARM MESSAGE);
When bit 1 (MCA) of parameter No. 6008 is set to 0:
The alarm screen displays "MC 3001 ALARM MESSAGE".
When bit 1 (MCA) of parameter No. 6008 is set to 1:
The alarm screen displays "MC0001 ALARM MESSAGE".

#3 KOP When the NC is reset in the state where the line is made open by POPEN:
0: Communication continues, and the line is left open.
1: Communication stops, and the line is closed.

#4 ISO 0: When the EIA code is used, the bit patterns of codes specified instead of [,], #, *, =, ?, @, &, and _ are set in parameters Nos. 6010 to 6018.
1: When the ISO/ASCII code is used, the bit patters of codes specified instead of [,], #, *, =, ?, @, &, and _ are set in parameters Nos. 6010 to 6018.

#5 ADD When the number of digits in the integer part, a, in the format specification [a,b] of the DPRNT statement is less than the number of digits in the integer part of an output variable value:
0: The specified number of digits only are output, with the unspecified digits discarded.
1: An alarm for excessive digits is issued.

#6 GMP The calling of M, S, T, a second auxiliary function code, or a particular code during the calling of a G code, and the calling of a G code during the calling of M, S, T, a second auxiliary function code, or particular code are:
0: Not allowed. (They are executed as an ordinary G, M, S, T, second auxiliary function code, and NC address.)
1: Allowed.

#7 IJK For addresses I, J, and K specified as arguments:
0: Argument specification I or II is automatically determined.
1: Argument specification I is always used.

Example

When K_J_I_ is specified:
- When this parameter is set to 0:
  Argument specification II is used and K=#6, J=#8, and I=#10 are specified.
- When this parameter is set to1:
  Argument specification I is used and I=#4, J=#5, and K=#6 are specified regardless of the specification order.
  (Argument specification II cannot be used.)

	#7	#6	#5	#4	#3	#2	#1	#0
6009						MAA		MSM

[Input type] Parameter input
[Data type] Bit path

#0 MSM When an M code or M codes specified for a macro call or macro calls (multiple macro call specification) are not at the beginning of the block:
0: Alarm PS0127”DUPLICATE NC,MACRO STATEMENT” is issued.
1: The macro or macros are called based on the M code or M codes. All addresses specified in the same block are used as arguments. (Special Macro Call using M code)

NOTE
1 When MSM is set to 1 and an M code specified for a macro call is not at the beginning of the block, argument specification II cannot be used.
2 When MSM is set to 1 and an M code specified for a macro call is not at the beginning of the block, the number of repetitions (L) cannot be used.

#2 MAA When a Macro Call using M code or Special Macro Call using M code are executed:
0: Address G does not become an argument.
The addresses that can be used as arguments are as shown in the tables below.
- Usual Macro Call

Address	Variable number
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
G	*1
H	#11
I	#4

Address	Variable number
J	#5
K	#6
L	*2
M	#13 *3
M(Call code)	*4
N	#14 *5
P	#16
Q	#17
R	#18

Address	Variable number
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

*1: Address G can not command. If address G is commanded, alarm PS0129“USE 'G' AS ARGUMENT ” occurs.
*2: Address L is a number of times in which the macro call is repeated.
*3: Address M other than call code is passed to variable #13.
*4: Address M for call code does not become an argument.
*5: Address N is passed to variable #14.

- Special Macro Call

Address	Variable number
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
G	*1
H	#11
I	#4

Address	Variable number
J	#5
K	#6
L	*2
M	#13 *3
M(Call code)	*4
N	#14 *5
P	#16
Q	#17
R	#18

Address	Variable number
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

*1: Even if address G is commanded, PS alarm does not occur. And address G does not become an argument.
*2: Address L does not become a number of times in which the macro call is repeated, and does not become an argument.
*3: Address M other than call code is passed to variable #13.
*4: Address M for call code does not become an argument.
*5: Address N is passed to variable #14, and becomes a sequence number.

1: Address G becomes an argument.The addresses that can be used as arguments are as shown in the tables below.

- Usual Macro Call

Address	Variable number
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
G	#28 to #32 *1
H	#11
I	#4

Address	Variable number
J	#5
K	#6
L	*2
M	#13 *3
M(Call code)	*4
N	#14 *5
P	#16
Q	#17
R	#18

Address	Variable number
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

*1: The first five addresses G in ascending order of G code groups are used as arguments and passed to variables #28 to #32.
*2: Address L is a number of times in which the macro call is repeated.
*3: Address M other than call code is passed to variable #13.
*4: Address M for call code does not become an argument.
*5: Address N is passed to variable #14.

- Special Macro Call

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	J	#5	S	#19
B	#2	K	#6	T	#20
C	#3	L	#12 *2	U	#21
D	#7	M	#13 *3	V	#22
E	#8	M(Call code)	#27 *4	W	#23
F	#9	N	#14 *5	X	#24
G	#28 to #32 *1	P	#16	Y	#25
H	#11	Q	#17	Z	#26
I	#4	R	#18		

*1: The first five addresses G in ascending order of G code groups are used as arguments and passed to variables #28 to #32.
*2: Address L is passed to variable #12, and does not become a number of times in which the macro call is repeated.
*3: Address M other than call code is passed to variable #13.
*4: Address M for call code is passed to variable #27.
*5: Address N is passed to variable #14, and becomes a sequence number.

	#7	#6	#5	#4	#3	#2	#1	#0
6010	*7	*6	*5	*4	*3	*2	*1	*0
6011	=7	=6	=5	=4	=3	=2	=1	=0
6012	#7	#6	#5	#4	#3	#2	#1	#0
6013	[7	[6	[5	[4	[3	[2	[1	[0
6014	]7	]6	]5	]4	]3	]2	]1	]0
6015	?7	?6	?5	?4	?3	?2	?1	?0
6016	@7	@6	@5	@4	@3	@2	@1	@0
6017	&7	&6	&5	&4	&3	&2	&1	&0
6018	_7	_6	_5	_4	_3	_2	_1	_0

[Input type] Parameter input
[Data type] Bit path

*0 to *7 : The bit pattern of the EIA or ISO/ASCII code indicating * is set.
=0 to =7 : The bit pattern of the EIA or ISO/ASCII code indicating = is set.
#0 to #7 : The bit pattern of the EIA or ISO/ASCII code indicating # is set.
[0 to [7 : The bit pattern of the EIA or ISO/ASCII code indicating [is set.
]0 to]7 : The bit pattern of the EIA or ISO/ASCII code indicating] is set.
?0 to ?7 : The bit pattern of the EIA or ISO/ASCII code indicating ? is set.
@0 to @7 : The bit pattern of the EIA or ISO/ASCII code indicating @ is set.
&0 to &7 : The bit pattern of the EIA or ISO/ASCII code indicating & is set.
_0 to _7 : The bit pattern of the EIA or ISO/ASCII code indicating _ is set.
0: A corresponding bit is 0.
1: A corresponding bit is 1.

	#7	#6	#5	#4	#3	#2	#1	#0
6019	SFN		EDP		OFN	DPD		
	SFN		EDP	MSV	OFN			

[Input type] Parameter input
[Data type] Bit

#2 DPD When argument D is specified for a macro call without a decimal point, the number of decimal places:
0: Is assumed to be 0.
[Example]
When G65PppppD1 is specified, #7=1.0000 is passed as the argument.
1: Depends on the increment system of the reference axis.
[Example]
When the increment system of the reference axis is IS-B and G65PppppD1 is specified, #7=0.0010 is passed as the argument.

#3 OFN The format of the name of a file output by the external output command (DPRNT or BPRNT) is:
0: PRNTxxxx.DAT (xxxx: 0000 to 9999).
1: MCR_PRNT.TXT (fixed).

#4 MSV When Tool length compensation shift type is used, the value in which Tool offset value, Tool length offset and Tool holder offset are :
#5041 - #5060, #100101 - #100150 (Current position)
#5061 - #5080, #100151 - #100200 (Skip position):
0: It is included in above-mentioned system value.
1: It is not included in above-mentioned system value.
Only in the machining center system, this parameter becomes effective. In the lathe system, bit 4 (TSV) of parameter No.6021 is valid. This parameter is valid both during coordinate system shift type compensation and axis movement type compensation if bit 4 (AXM) of parameter No.6005 is set to 1.

#5 EDP Precision setting for macro relational operators is:
0: Disabled.
1: Enabled.

NOTE
Parameter No. 6100 is used to set the number of significant digits after the decimal point.

#7 SFN The format of the name of a file output by the external output command (DPRNT or BPRNT) is:
0: PRNTxxxx.DAT (xxxx: 0000 to 9999).
If the power is off and on, xxxx is reset to 0000.
1: PRNTxxxx.DAT (xxxx: 0000 to 9999).
If the power is off and on, xxxx is continued and the next number is applied.

NOTE
The setting value of bit 7 (SFN) of parameter No. 6019 is effective in case of that bit 3 (OFN) of parameter No. 6019 equals 0.

	#7	#6	#5	#4	#3	#2	#1	#0
6020					NCM	IFR	NC2	NC1

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 NC1 The setting of the number of custom macro variables common to paths for #100 to #199 (#499) (parameter No. 6036) is:

0: Valid.
As many custom macro variables #100 to #199 (or #100 to #499) set for this path as the number set in parameter No. 6036 are used as custom macro variables common to tool paths.

1: Invalid.
Custom macro variables #100 to #199 (or #100 to #499) set for this path are all used as variables for the path.

> **NOTE**
> For path 1, be sure to set this parameter to 0.

> **Example**
> In a 4-path system, when parameters are set as listed below, custom macro variables for paths 1 to 3 are used as variables common to these paths, but for path 4, custom macro variables for the path are used.

Path number	No. 6036	NC1	Used custom macro variables
1	20	0	Custom macro variables #100 to #119 are used as variables common to these paths, and other custom macro variables are used independently for the relevant path.
2		0	
3		0	
4		1	Custom macro variables for path 4 are all used independently.

#1 NC2 The setting of the number of custom macro variables common to paths for #500 to #999 (parameter No. 6037) is:

0: Valid.
As many custom macro variables #500 to #999 set for this path as the number set in parameter No. 6037 are used as custom macro common variables between each path.

1: Invalid.
Custom macro variables #500 to #999 set for this path are all used as variables for the path.

> **NOTE**
> For path 1, be sure to set this parameter to 0.

Example

In a 4-path system, when parameters are set as listed below, custom macro variables for paths 1 to 3 are used as variables common to these paths, but for path 4, custom macro variables for the path are used.

Path number	No. 6037	NC2	Used custom macro variables
1	50	0	Custom macro variables #500 to #999 are used as variables common to these paths, and other custom macro variables are used independently for the relevant path.
2		0	
3		0	
4		1	Custom macro variables for path 4 are all used independently.

#2 IFR The custom macro interface signal R address is:
0: Disabled.
1: Enabled.

NOTE

Set the start address of each R area in parameter No. 6093 or 6094.

⚠ **WARNING**

To set the parameter IFR=1, set an appropriate value to parameter No.6094 beforehand.
When the parameter No.6094 is 0, R0~ of an internal relay is set. If the internal relay R0~ is used to another area, the operation not intended might be done.

#3 NCM Position where comment section of machining program can be inserted and handling of comment section inserted in block of #3000 (macro alarm) / #3006 (stop with a message) are decided.

0: The position in which comment section can be inserted in macro statements is as follows.

(ABC) #100 =1; Head of the block
#100 =1 (ABC); End of the block
N01 (ABC) #100 =1; Just behind of the sequence number
It cannot be inserted just after variable / constant value / variable name.

When inserting the comment section into the block of # 3000 (macro alarm) / # 3006 (macro message), the order of the character string as the message and the character string as comment section part is as follows.

#3000 =1 (ALARM MESSAGE) (COMMENT 1) (COMMENT 2);

The first bracket is assumed as the message, and the second and subsequent ones as the comment section.

1: The position in which comment section can be inserted in macro statements is as follows.

(ABC) #100 =1; Head of the block
#100 =1 (ABC); End of the block
N01 (ABC) #100 =1; Just behind of the sequence number
#100(ABC) =1; Just behind of variable number
#100 =#101 +1.(ABC) *#102; Just behind of numeric values
#100 =[#_UIL[1]](ABC) *100.; Just behind of the name of variable

When inserting the comment section into the block of # 3000 (macro alarm) / # 3006 (macro message), the order of the character string as the message and the character string as comment section part is as follows.

#3000 =1 (COMMENT 1) (COMMENT 2) (ALARM MESSAGE);

The last bracket is regarded as a message, and brackets before it are regarded as an comment section.

	#7	#6	#5	#4	#3	#2	#1	#0
6021				TSV				

[Input type] Parameter input
[Data type] Bit

#4 TSV When Tool length compensation shift type is used, the value in which Tool offset value, Tool length offset and Tool holder offset are :
#5041 - #5060, #100101 - #100150 (Current position)
#5061 - #5080, #100151 - #100200 (Skip position):
0: It is included in above-mentioned system value.
1: It is not included in above-mentioned system value.
Only in the lathe system, this parameter becomes effective. In the machining center system, bit 4 (MSV) of parameter No.6019 is valid. This parameter is valid both during coordinate system shift type compensation and axis movement type compensation if bit 4 (AXM) of parameter No.6005 is set to 1.

NOTE
1 If bit 2 (LWT) of parameter No.5002 is set to 1 (coordinate system shift type wear compensation) and tool compensation is executed, the wear compensation value is not included in the system variables independently of bit 4 (TSV) of parameter No.6021.
2 If bit 4 (LGT) of parameter No.5002 is set to 0 (coordinate system shift type geometry compensation) and tool compensation is executed, the geometry compensation value is not included in the system variables independently of bit 4 (TSV) of parameter No.6021.

6031	Start number of common variables to be protected among the common variables (#500 to #999)
6032	End number of common variables to be protected among the common variables (#500 to #999)

[Input type] Parameter input
[Data type] Word path
[Valid data range] 500 to 999

Among the common variables #500 to #999, the range of common variables specified by this parameter can be protected (by setting their attributes to read-only). If a write attempt (on the left side) is made, an alarm is issued.

NOTE
Set 0 in both parameters Nos. 6031 and 6032 not to protect common variables.

6036	Number of custom macro variables common between each path (for #100 to #199 (#499))

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 400

When the memory common to paths is used, this parameter sets the number of custom macro common variables to be shared (custom macro variables common to paths). Common variables #100 to #199 (up to #499 in a system with the embedded macro option) may be shared. Ensure that the maximum number of usable macro common variables is not exceeded.

Example
When 20 is set in parameter No. 6036
#100 to #119: Shared by all paths
#120 to #149: Used by each path independently

NOTE
1 When the adding custom macro common variables is enabled (bit 6(NCV) of parameter No.8135 is 0), up to #199 is available.
2 To use up to #499, the embedded macro option is required.
3 When 0 or a negative value is set, the memory common to paths is not used.
4 When the option for embedded macro is effective and the addition of custom macro common variables is disabled (bit 6 (NCV) of parameter No.8135 is 1), #150 to #199 can not be used but this parameter should be set the number which includes #150 to #199.

6037	Number of custom macro variables common between each path (for #500 to #999)

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 500

When the memory common to paths is used, this parameter sets the number of custom macro common variables to be shared (custom macro variables common to paths). Common variables #500 to #999 may be shared. Ensure that the maximum number of usable macro common variables is not exceeded.

Example
When 50 is set in parameter No. 6037
#500 to #549: Shared by all paths
#550 to #599: Used by each path independently

NOTE
1 When the adding custom macro common variables is enabled (bit 6(NCV) of parameter No.8135 is 0), up to #999 is available.
2 When 0 or a negative value is set, the memory common to paths is not used.

6038	Start G code used to call a custom macro

[Input type] Parameter input
[Data type] Word path

[Valid data range] -9999 to 9999

6039	Start program number of a custom macro called by G code

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

6040	Number of G codes used to call custom macros

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 255

Set this parameter to define multiple custom macro calls using G codes at a time. With G codes as many as the value set in parameter No. 6040 starting with the G code set in parameter No. 6038, the custom macros of program numbers as many as the value set in parameter No. 6040 starting with the program number set in parameter No. 6039 can be called. Set 0 in parameter No. 6040 to disable this mode of calling.

If a negative value is set in parameter No. 6038, the modal call mode is entered. Whether the modal call is equivalent to G66 or G66.1 depends on bit 3 (MGE) of parameter No. 6007.

[Example] When parameter No. 6038 = 900, parameter No. 6039 = 1000, and parameter No. 6040 = 100 are set, a set of 100 custom macro calls (simple calls) is defined as follows:

G900 → O1000
G901 → O1001
G902 → O1002
:
G999 → O1099

When the setting of parameter No. 6038 is changed to -900, the same set of custom macro calls (modal calls) is defined.

NOTE

1. When the following conditions are satisfied, all calls using these parameters are disabled:
   1) When a value not within the specifiable range is set in each parameter
   2) (Value of parameter No. 6039 + value of parameter No. 6040 - 1) > 9999
2. The specification of a mixture of simple calls and modal calls is not allowed.
3. If a range of G codes set by these parameters duplicate G codes specified in parameters Nos. 6050 to 6059, the calls specified by parameters Nos. 6050 to 6059 are made preferentially.

6041	Start G code with a decimal point used to call a custom macro

[Input type] Parameter input
[Data type] Word path
[Valid data range] -999 to 999

6042	Start program number of a custom macro called by G code with a decimal point

[Input type] Parameter input

[Data type] 2-word path
[Valid data range] 1 to 9999

6043	Number of G codes with a decimal point used to call custom macros

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 255

Set this parameter to define multiple custom macro calls using G codes with a decimal point at a time. With G codes with a decimal point as many as the value set in parameter No. 6043 starting with the G code with a decimal point set in parameter No. 6041, the custom macros of program numbers as many as the value set in parameter No. 6043 starting with the program number set in parameter No. 6042 can be called. Set 0 in parameter No. 6043 to disable this mode of calling.

If a negative value is set in parameter No. 6041, the modal call mode is entered. Whether the modal call is equivalent to G66 or G66.1 depends on bit 3 (MGE) of parameter No. 6007.

[Example] When parameter No. 6041 = 900, parameter No. 6042 = 2000, and parameter No. 6043 = 100 are set, a set of 100 custom macro calls (simple calls) is defined as follows:

G90.0 → O2000
G90.1 → O2001
G90.2 → O2002
:
G99.9 → O2099

When the setting of parameter No. 6041 is changed to -900, the same set of custom macro calls (modal calls) is defined.

NOTE

1 When the following conditions are satisfied, all calls using these parameters are disabled:
   1) When a value not within the specifiable range is set in each parameter
   2) (Value of parameter No. 6042 + value of parameter No. 6043 - 1) > 9999
   3) When bit 0 (DPG) of parameter No. 6007 = 0 (to disable calls using G codes with a decimal point)
2 The specification of a mixture of simple calls and modal calls is not allowed.
3 If a range of G codes set by these parameters duplicate G codes specified in parameters Nos. 6060 to 6069, the calls specified by parameters Nos. 6060 to 6069 are made preferentially.

6044	Start M code used to call a subprogram

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999

6045	Start program number of a subprogram called by M code

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

6046	Number of M codes used to call subprograms (number of subprograms called by M codes)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 32767

Set this parameter to define multiple subprogram calls using M codes at a time. With M codes as many as the value set in parameter No. 6046 starting with the M code set in parameter No. 6044, the subprograms of program numbers as many as the value set in parameter No. 6046 starting with the program number set in 6045 can be called. Set 0 in parameter No. 6046 to disable this mode of calling.

[Example] When parameter No. 6044 = 80000000, parameter No. 6045 = 3000, and parameter No. 6046 = 100 are set, a set of 100 subprogram calls is defined as follows:

M80000000 → O3000
M80000001 → O3001
M80000002 → O3002
:
M80000099 → O3099

NOTE

1. When the following conditions are satisfied, all calls using these parameters are disabled:
   1) When a value not within the specifiable range is set in each parameter
   2) (Value of parameter No. 6045 + value of parameter No. 6046 - 1) > 9999
2. If a range of M codes set by these parameters duplicate M codes specified in parameters Nos. 6071 to 6079, the calls specified by parameters Nos. 6071 to 6079 are made preferentially.

6047	Start M code used to call a custom macro

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999

6048	Start program number of a custom macro called by M code

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

6049	Number of M codes used to call custom macros (number of custom macros called by M codes)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 32767

Set this parameter to define multiple custom macro calls using M codes at a time. With M codes as many as the value set in parameter No. 6049 starting with the M code set in parameter No. 6047, the custom macros of program numbers as many as the value set in parameter No. 6049 starting with the program number set in parameter No. 6048 can be called. Set 0 in parameter No. 6049 to disable this mode of calling.

[Example] When parameter No. 6047 = 90000000, parameter No. 6048 = 4000, and parameter No. 6049 = 100 are set, a set of 100 custom macro calls (simple calls) is defined as follows:
M90000000 → O4000
M90000001 → O4001
M90000002 → O4002
:
M90000099 → O4099

> **NOTE**
> 1 When the following conditions are satisfied, all calls using these parameters are disabled:
> 1) When a value not within the specifiable range is set in each parameter
> 2) (Value of parameter No. 6048 + value of parameter No. 6049 - 1) > 9999
>
> 2 If a range of M codes set by these parameters duplicate M codes specified in parameter No. 6080 through No. 6089, the calls specified by parameter No. 6080 through 6089 are made preferentially.

6050	**G code that calls the custom macro of program number 9010**
to	**to**
6059	**G code that calls the custom macro of program number 9019**

[Input type] Parameter input
[Data type] Word path
[Valid data range] (-9999 to 9999 : excluding 0, 5, 65, 66 and 67)

Set the G codes used to call the custom macros of program numbers 9010 to 9019. However, note that when a negative value is set in this parameter, it becomes a modal call. For example, if this parameter is set to -11, the modal call mode is entered by G11.
Whether the modal call is equivalent to G66 or G66.1 depends on bit 3 (MGE) of parameter No. 6007.

6060	**G code with a decimal point used to call the custom macro of program number 9040**
to	**to**
6069	**G code with a decimal point used to call the custom macro of program number 9049**

[Input type] Parameter input
[Data type] Word path
[Valid data range] -999 to 999

Set the G codes used to call the custom macros of program numbers 9040 to 9049. However, note that when a negative value is set in this parameter, it becomes a modal call. For example, if this parameter is set to -11, the modal call mode is entered by G11.
Whether the modal call is equivalent to G66 or G66.1 depends on bit 3 (MGE) of parameter No. 6007. Set G codes in the format Gm.n. The value expressed by ($m \times 10+n$) is set in the parameter. The values m and n must satisfy the following relationships: $0 \leq m \leq 99$, $0 \leq n \leq 9$.

> **NOTE**
> Parameter Nos. 6060 to 6069 are valid when bit 0 (DPG) of parameter No. 6007 is set to 1.

6071	**M code used to call the subprogram of program number 9001**
to	**to**
6079	**M code used to call the subprogram of program number 9009**

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999 (excluding 30, 98 and 99)

These parameters set the M codes that call the subprograms of program numbers 9001 to 9009.

> **NOTE**
> If the same M code is set in these parameters, the younger number is called preferentially. For example, if 100 is set in parameter No. 6071 and 6072, and programs O9001 and O9002 both exist, O9001 is called when M100 is specified.

6080	**M code used to call the custom macro of program number 9020**
to	**to**
6089	**M code used to call the custom macro of program number 9029**

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999 (excluding 30, 98 and 99)

Set the M codes used to call the custom macros of program numbers 9020 to 9029. The simple call mode is set.

> **NOTE**
> 1 If the same M code is set in these parameters, the younger number is called preferentially. For example, if 200 is set in parameters Nos. 6081 and 6082, and programs O9021 and O9022 both exist, O9021 is called when M200 is specified.
> 2 If the same M code is set in a parameters Nos. 6071 to 6079 used to call subprograms and in a parameters Nos. 6080 to 6089 used to call custom macros, a custom macro is called preferentially. For example, if 300 is set in parameters Nos. 6071 and 6081, and programs O9001 and O9021 both exist, O9021 is called when M300 is specified.

6090	**ASCII code that calls the subprogram of program number 9004**
6091	**ASCII code that calls the subprogram of program number 9005**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 65(A:41H) to 90(Z:5AH)

These parameters set the ASCII codes that call subprograms in decimal.
The settable addresses are indicated below.

Address	Parameter setting value	T series	M series
A	65	O	O
B	66	O	O
D	68	X	O
F	70	O	O

Address	Parameter setting value	T series	M series
H	72	O	O
I	73	O	O
J	74	O	O
K	75	O	O
L	76	O	O
M	77	O	O
P	80	O	O
Q	81	O	O
R	82	O	O
S	83	O	O
T	84	O	O
V	86	X	O
X	88	X	O
Y	89	X	O
Z	90	X	O

NOTE
1 When address L is set, the number of repeats cannot be specified.
2 Set 0 when no subprogram is called.

6093	Top address of custom macro interface signal R address (input signal)

6094	Top address of custom macro interface signal R address (output signal)

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to maximum address (multiple of 4. 0, 4, 8, ...)
Set the top address of custom macro interface signal R address. 128 signals starting at the top address are allocated.

[Example]

Parameter	System variable number	Signal to use	Attribute
No.6093=1000	#1068 #1069 #1070 #1071	R1000 to R1003 R1004 to R1007 R1008 to R1011 R1012 to R1015	R
No.6094=1100	#1168 #1169 #1170 #1171	R1100 to R1103 R1104 to R1107 R1108 to R1111 R1112 to R1115	R/W

*) The R and R/W attributes of variables represent, respectively, read-only and read/write enabled.

NOTE
1. Each value specified with this parameter must be a multiple of 4 (0, 4, 8, ⋯). Otherwise, this function is disabled.
2. If a non-existent R address or system relay address is set, the corresponding system variable becomes invalid. The effective R address area varies depending on the PMC used and its memory. Be sure to specify a usable range by checking the specification of the PMC in use.
3. When setting the parameter, make sure that the input signal addresses do not overlap with the output signal addresses.

6100	Precision setting for relational operators

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 15

This parameter sets the number of digits after the decimal point in two values to be compared using the custom macro relational operator. The two values are rounded off to the specified number of digits before comparison.

NOTE
1. This function is enabled by setting bit 5 (EDP) of parameter No. 6019 to 1.
2. This function is disabled if parameter No. 6100 is set to a value out of the valid data range.

	#7	#6	#5	#4	#3	#2	#1	#0
6210						DSK		

[Input type] Parameter input
[Data type] Bit path

#2 DSK Skip position reading (system variables #5421 to #5440, #100701 to #100750) by the detection unit is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
8106						CVP		

[Input type] Parameter input
[Data type] Bit

 # 2 CVP By a reset in path other than path 1, the custom macro variables common to paths set to the common variables #100 to #499 cleared by power-off is
0: not cleared to <null>.
1: cleared to <null>. (it depend to the bit 6(CCV) of parameter No.6001.)

> **NOTE**
> 1 When a reset is made in path 1, it is cleared regardless of the setting of this parameter. (it depend to the parameter CCV)
> 2 This parameter is invalid for the path that the bit 0(NC1) of parameter No.6020 is 1.

	#7	#6	#5	#4	#3	#2	#1	#0
11648						ESE		

[Input type] Parameter input
[Data type] Bit

#2 ESE ELSE statement of IF statement is:
0: Not used
1: Used. The following specifications are added.
- The following instruction format is added to IF statement.
  - IF[<conditional expression>] THEN; ~ ENDIF;
  - IF[<conditional expression>] THEN Macro statement1;
    ELSE Macro statement2;
  - IF[<conditional expression>] THEN; ~ ELSE; ~ ENDIF;
- IF statement multiplexing (nest management) is enable.
- "ELSE", and "ENDIF" are added to reserved words.

11649	Number of custom macro variables common to each path (for #98000 to #98499)

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 500

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

When the custom macro variables common to each path is used, this parameter sets the number of custom macro common variables to be shared out of common variables #98000 to #98499. Be sure that the maximum number of available macro common variables is not exceeded.

> **Example**
> When 100 is set in parameter No.11649
> #98000 to #98099: Variables are shared by all paths
> #98100 to #98499: Variables are used by each path independently

> **NOTE**
> 1 Addition of custom macro common variables 1000 is required.
> 2 When 0 or a negative value is set, the custom macro variables common to each path is not used.
> 3 By parameter NC4(No.11654#0), commonness/independence of variables can be set for each path.

11652	Start number of common variables to be protected among the common variables (#98000 to #98499)

11653	End number of common variables to be protected among the common variables (#98000 to #98499)

[Input type] Parameter input
[Data type] 2-Word path
[Valid data range] 0, 98000 to 98499

Among the common variables #98000 to #98499, the range of common variables specified by this parameter can be protected (by setting their attributes to read-only). If value is written to protected variable (on the left side), an alarm is issued.

NOTE
Set 0 in both parameter No.11652 and No.11653 not to protect common variables.

	#7	#6	#5	#4	#3	#2	#1	#0
11654								NC4

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

 # 0 NC4 The parameter No.11649 (Number of custom macro variables common to each path for #98000 to #98499) is:
0: Effective.
Among custom macro variable #98000-#98499 in this path, the number set in parameter No.11649 becomes the number of variables shared between each path.
1: Not effective.
Custom macro variable #98000-#98499 in this path are independent of other path.

NOTE
This parameter for path 1 must be set to 0.

Example
In a 4-path system, when parameters are set as listed below, custom macro variables #98000 to #98099 for paths 1 to 3 are used as variables common to these paths, but for path 4, custom macro variables #98000 to #98099 for the path 4 are used independently.

Path number	No.11649	NC4	Used custom macro variables
1	100	0	Custom macro variables #98000 to #98099 are used as variables common to these paths, and other custom macro variables are used independently.
2		0	
3		0	
4		1	All custom macro variables for path 4 are used independently.

Alarm and message

Number	Message	Description
PS0077	TOO MANY SUB,MACRO NESTING	The total number of subprogram and macro calls exceeds the permissible range. Another subprogram call was executed during an external memory subprogram call.
PS0110	OVERFLOW :INTEGER	An integer went out of range during arithmetic calculations.
PS0111	OVERFLOW :FLOATING	A decimal point (floating point number format data) went out of range during arithmetic calculations.
PS0112	ZERO DIVIDE	An attempt was made to divide by zero in a custom macro.
PS0114	ILLEGAL EXPRESSION FORMAT	The format used in an expression in a custom macro statement is in error. The parameter program format is in error.
PS0115	VARIABLE NO. OUT OF RANGE	An illegal No. was specified in a local variable, common variable or a system variable in a custom macro. A non–existent custom macro variable No. was specified in the EGB axis skip function (G31.8), or there are not enough custom macro variables for storing the skip position.
PS0116	WRITE PROTECTED VARIABLE	An attempt was made in a custom macro to use on the left side of an expression a variable that can only be used on the right side of an expression.
PS0118	TOO MANY BRACKET NESTING	Too many brackets "[ ]" were nested in a custom macro. Nesting of [] is up to five times including [] of function.
PS0119	ARGUMENT VALUE OUT OF RANGE	The value of an argument in a custom macro function is out of range.
PS0122	TOO MANY MACRO NESTING	The nesting of the custom macro call exceeded the allowable range.
PS0124	MISSING END STATEMENT	The END instruction corresponding to the DO instruction was missing in a custom macro.
PS0125	MACRO STATEMENT FORMAT ERROR	The format used in a macro statement in a custom macro is in error.
PS0126	ILLEGAL LOOP NUMBER	There is an error in the number of the DO and END statements of the custom macro, or it exceeds the allowable range (1 to 3).
PS0127	DUPLICATE NC,MACRO STATEMENT	An NC statement and macro statement were specified in the same block.
PS0129	USE 'G' AS ARGUMENT	G is used as an argument in a custom macro call. G can be specified as an argument only in an every-block call (G66.1).
PS1091	DUPLICATE SUB-CALL WORD	More than one subprogram call instruction was specified in the same block.
PS1092	DUPLICATE MACRO-CALL WORD	More than one macro call instruction was specified in the same block.
PS1093	DUPLICATE NC-WORD & M99	An address other than O, N, P or L was specified in the same block as M99 during the macro modal call state.
PS1095	TOO MANY TYPE-2 ARGUMENT	More than ten sets of I, J and K arguments were specified in the type–II arguments (A, B, C, I, J, K, I, J, K, ...) for custom macros.
PS1096	ILLEGAL VARIABLE NAME	An illegal variable name was specified. A code that cannot be specified as a variable name was specified. [#_OFSxx] does not match the tool offset memory option configuration.
PS1097	TOO LONG VARIABLE NAME	The specified variable name is too long.
PS1098	NO VARIABLE NAME	The specified variable name cannot be used as it is not registered.

Number	Message	Description
PS1099	ILLLEGAL SUFFIX [ ]	A suffix was not specified to a variable name that required a suffix enclosed by [ ]. A suffix was specified to a variable name that did not require a suffix enclosed by [ ]. The value enclosed by the specified [ ] was out of range.
PS1100	CANCEL WITHOUT MODAL CALL	Call mode cancel (G67) was specified even though macro continuous–state call mode (G66) was not in effect.
PS1101	ILLEGAL CNC STATEMENT IRT.	An interrupt was made in a state where a custom macro interrupt containing a move instruction could not be executed.
PS1115	READ PROTECTED VARIABLE	An attempt was made in a custom macro to use on the right side of an expression a variable that can only be used on the left side of an expression.
PS1124	MISSING DO STATEMENT	The DO instruction corresponding to the END instruction was missing in a custom macro.
PS1128	SEQUENCE NUMBER OUT OF RANGE	The jump destination sequence No. in a custom macro statement GOTO instruction was out of range (valid range: 1 to 99999999).
PS1129	MISSING IF STATEMENT	There is no “IF” corresponding to "ELSE" and "ENDIF".
PS1130	TOO MANY IF STATEMENT NESTING	The maximum value of the nesting of IF statement was exceeded. Up to 10 levels are allowed.
PS1131	MISSING OPEN BRACKET	The number of left brackets ([) is less than the number of right brackets ()) in a custom macro statement.
PS1132	MISSING CLOSE BRACKET	The number of right brackets (]) is less than the number of left brackets ([] in a custom macro statement.
PS1133	MISSING '='	An assignment directive (=) is missing in the arithmetic calculation instruction in a custom macro statement.
PS1137	IF STATEMENT FORMAT ERROR	The format used in the IF statement in a custom macro is in error.
PS1138	WHILE STATEMENT FORMAT ERROR	The format used in the WHILE statement in a custom macro is in error.
PS1139	SETVN STATEMENT FORMAT ERROR	The format used in the SETVN statement in a custom macro is in error.
PS1141	ILLEGAL CHARACTER IN VAR. NAME	The SETVN statement in a custom macro contacts a character that cannot be used in a variable name.
PS1142	TOO LONG V-NAME (SETVN)	The variable name used in a SETVN statement in a custom macro exceeds 8 characters (31 characters when the custom macro variable name expansion function is enabled).
PS1143	BPRNT/DPRNT STATEMENT FORMAT ERROR	The format used in the BPRINT statement or DPRINT statement is in error.

Diagnosis data

Diagnosis	1493	Number of macro statement blocks executed by custom and execution macros

[Data type] 2-word

[Unit of data] Block

The number of macro statement blocks executed by custom and execution macros per 1024 ms is displayed.

This data provides an indication of the actual macro statement processing speed.

12.6.2 Indirect Axis Address Command

Overview

When the custom macro function is enabled, indirect specification by an axis number can be performed by using AX[(axis-number)] instead of directly specifying an axis name during specification for an axis address.
In addition, the axis number corresponding to the axis name can be obtained by using AXNUM[(axis-name)].

Explanation

- Indirect axis address

Indirect axis address AX[] can be used to perform a command for an axis with an axis number (AX[] must be followed by "=").

AX[(axis-number)]=(value);

(axis-number) :1 to number-of-control-axis (Number of each path for a multipath system)
(value) :Value for the axis specified by its axis number

When the specified (axis-number) is invalid, the alarm PS0331, "ILLEGAL AXIS NUMBER IN AX[]" occurs. If there is a value in the decimal place digits, the number rounded off is treated as (axis number).
A variable such as a local variable, common variable, or system variable can also be used as (axis-number). When calculation using a variable name is performed for (axis-number), however, the variable name must be enclosed with [].

1. AX[1]=100.0;
   A value of 100.000 is specified for the first axis.
2. AX[#500]=200.0;
   A value of 200.000 is specified for the axis having the value stored in #500 as its axis number.
3. AX[#500+1]=300.0;
   A value of 300.000 is specified for the axis having the value stored in #500 plus 1 as its axis number.
4. SETVN 500 [ABC];
   AX[#ABC]=400.0;
   A value of 400.000 is specified for the axis having the value stored in #ABC(#500) as its axis number.
5. SETVN 500 [ABC];
   AX[[#ABC]+1]=500.0;
   A value of 500.000 is specified for the axis having the value stored in #ABC(#500) plus 1 as its axis number.
6. SETVN 500 [ABC];
   AX[#ABC+1]=500.0;
   The alarm PS0114"ILLEGAL EXPRESSION FORMAT" occurs.

- AXNUM function

AXNUM[] can be used to obtain an axis number.

AXNUM[(axis-name)];

If the specified (axis-name) is invalid, the alarm PS0332, "ILLEGAL AXIS ADDRESS IN AXNUM[]" occurs.

When the number of control axes is three and the first axis name is "X", the second axis name is "Y", and the third axis name is "Z"
1. #500=AXNUM[X];
   A value of 1 is stored in #500.
2. #501=AXNUM[Y];
   A value of 2 is stored in #501.
3. #502=AXNUM[Z];
   A value of 3is stored in #502.
4. #503=AXNUM[A];
   The alarm PS0332 occurs.

Example

Example where the first axis name is "X", the second axis name is "Y", and the third axis name is "Z1"

```
N10 SETVN 500[AXIS1,AXIS2,AXIS3] ;
N20 [#AXIS1]=AXNUM[X] ;
N30 [#AXIS2]=AXNUM[Y] ;
N40 [#AXIS3]=AXNUM[Z1] ;
N50 G92 AX[#AXIS1]=0 AX[#AXIS2]=0 AX[#AXIS3]=0 ;
N60 G01F1000. ;
N70 AX[#AXIS1]=100.0 AX[#AXIS2]=100.0 AX[#AXIS3]=100.0 ;
N80 G02 AX[#AXIS1]=200. 0 AX[#AXIS1]=200.0 R50.0 ;
N90 M02;
```

Limitation

When the custom macro function is enabled, "AX" and "AXN" cannot be used as extended axis names and are assumed as AX[] and AXNUM[], respectively.

Alarm and message

Number	Message	Description
PS0331	ILLEGAL AXIS NUMBER IN AX[]	An illegal value is specified for an AX[] axis number.
PS0332	ILLEGAL AXIS ADDRESS IN AXNUM[]	An illegal value is specified for an AXNUM[] axis address.

12.6.3 Interruption Type Custom Macro

Overview

When a program is being executed, another program can be called by inputting an interrupt signal UINT <Gn053.3> from the machine. This function is referred to as an interruption type custom macro function.

Explanation

Program an interrupt command in the following format:

M96 Pxxxx;	**Enables custom macro interrupt**
M97 ;	**Disables custom macro interrupt**

Use of the interruption type custom macro function allows the user to call a program during execution of an arbitrary block of another program. This allows programs to be operated to match situations which vary from time to time.

(1) When a tool abnormality is detected, processing to handle the abnormality is started by an external signal.

(2) A sequence of machining operations is interrupted by another machining operation without the cancellation of the current operation.
(3) At regular intervals, information on current machining is read.
:

Listed above are examples like adaptive control applications of the interruption type custom macro function.

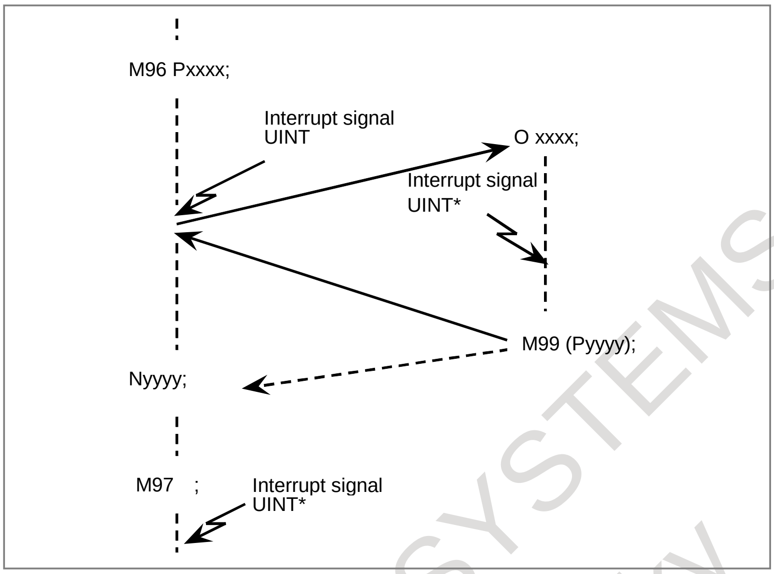


Fig. 12.6.3 Interruption type custom macro function

When M96Pxxxx is specified in a program, subsequent program operation can be interrupted by an interrupt signal UINT input to execute the program specified by Pxxxx. When the interrupt signal UINT (marked with an asterisk (*) in Fig. 12.6.3) is input during execution of the interrupt program or after M97, it is ignored.

Signal

Interrupt signal for custom macro UINT<Gn053.3>

[Classification] Input signal

[Function] This signal calls and executes a program in memory.
During execution, a program in automatic operation is suspended.
To enable this signal to be accepted, a particular miscellaneous function must be specified in a command program for automatic operation. In addition, automatic operation must already be started to accept this signal. The particular miscellaneous function code is set by parameter Nos. 6003, 6033 and 6034.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn053					UINT			

Parameter

- Various Setting for Custom Macro

	#7	#6	#5	#4	#3	#2	#1	#0
6003	MUS		MSB	MPR	TSE	MIN	MSK	

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#1 MSK Absolute coordinates at that time during custom macro interrupt
- 0: Not set to the skip coordinates (system variables #5061 and later)
- 1: Set to the skip coordinates (system variables #5061 and later)

#2 MIN Custom macro interrupt
- 0: Performed by interrupting an in-execution block (Custom macro interrupt type I)
- 1: Performed after an in-execution block is completed (Custom macro interrupt type II)

#3 TSE Custom macro interrupt signal UINT
- 0: Edge trigger method (Rising edge)
- 1: Status trigger method

#4 MPR Custom macro interrupt valid/invalid M code
- 0: M96/M97
- 1: M code set using parameters Nos. 6033 and 6034

#5 MSB Interrupt program
- 0: Uses a dedicated local variable (Macro-type interrupt)
- 1: Uses the same local variable as in the main program (Subprogram- type interrupt)

#7 MUS Interrupt-type custom macro
- 0: Not used
- 1: Used

- Setting M code that makes custom macro interrupt effective/ineffective

6033	**M code that validates a custom macro interrupt**

6034	**M code that invalidates a custom macro interrupt**

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 03 to 99999999 (excluding 30, 98 and 99)

These parameters can be used when bit 4 (MPR) of parameter No. 6003 is 1. M96 is used as a valid M code and M97 is used as an invalid M code when MPR is 0, irrespective of the state of this parameter.

Note

> **NOTE**
> 1 No interrupt-type custom macro can be used during multiple repetitive canned cycle execution.
> 2 No interrupt-type custom macro can be used during return operation in dry run after search operation when the program is restarted.

NOTE

3 The alarm PS1101 is issued in the following cases:
- <1> Case where an interrupt is made in the programmable mirror image (G51.1) mode and G51.1 is further specified in the interrupted program
- <2> Case where an interrupt is made in the coordinate system rotation (G68) mode and G68 is further specified in the interrupted program
- <3> Case where an interrupt is made in the scaling (G51) mode and G51 is further specified in the interrupted program

4 During program execution in cycle operation, interrupt type II is used, regardless of the setting of bit 2 (MIN) of parameter No. 6003. Cycle operation has the following functions:
- <1> Automatic reference position return
- <2> Cutter or tool nose radius compensation (when multiple blocks are generated from one specified block as in a case where an acute turn is made on the outside)
- <3> Canned cycle
- <4> Automatic tool length measurement (M series) / Automatic tool offset (T series)
- <5> Optional chamfering corner R
- <6> Normal direction control
- <7> Cutting point interpolation for cylindrical interpolation

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Interrupt-type custom macro

12.6.4 Embedded Macro

Overview

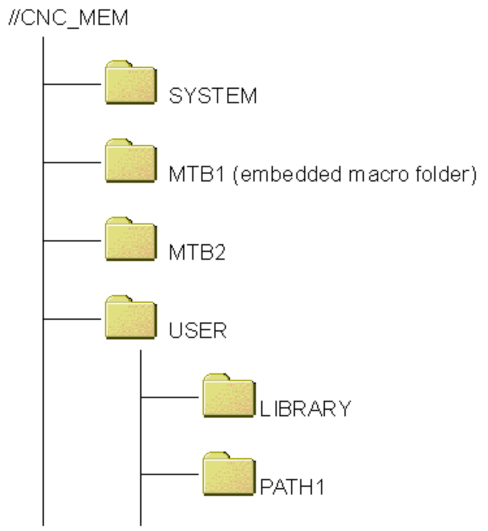
This function protects a program created by a machine tool builder, by storing the program in a folder dedicated to embedded macros (hereinafter referred to as an MTB1 folder) and assigning an attribute to the folder.
At the same time, the following functions are available:

(1) A dedicated program memory capacity of 100K bytes (corresponding to about 260 m) is provided in addition the ordinary program memory capacity.
The number of registerable programs remains unchanged from the ordinary number of registerable programs.

(2) To the MTB1 folder, an attribute such as edit prohibition and edit/display prohibition can be assigned. Moreover, by using a dedicated password and keyword, a set value can be locked.

(3) A program stored in the MTB1 folder can be used by calling based on a code such as M/T codes, macro calling based on a G code, macro calling based on G65/G66, and subprogram calling based on an M code such as M98.

(4) Custom macro common variables (#200 to #499) are added.



Detailed explanation

(1) Program memory capacity

A dedicated program memory capacity of 100K bytes (corresponding to about 260 m) is provided in addition the ordinary program memory capacity.

The dedicated program memory capacity is managed separately from the ordinary program memory capacity. Program memory capacity information (used pages and free pages) is displayed as described below.

- When the MTB1 folder is displayed
  Embedded macro program memory capacity
- When another folder is displayed
  Ordinary program memory capacity

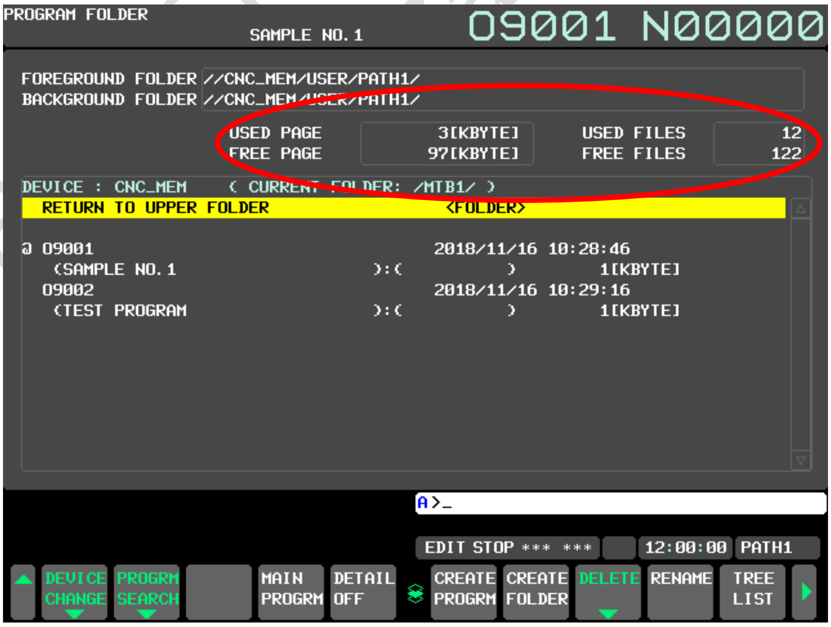


Fig. 12.6.4 (a) Program memory capacity

(2) Number of registerable programs

Ordinary programs and dedicated programs can be registered as long as the total number of those programs does not exceed the number of registerable programs.

(3) Program protection function
One of two attributes can be assigned to the MTB1 folder.
1) Edit prohibition
Editing of the programs in the folder is prohibited.
- The contents of programs cannot be edited.
- No program name and program attribute can be modified.
- No program can be deleted.
No new program can be created in the folder.
2) Edit/display prohibition
The contents of programs in the folder cannot be edited.
No new program can be created in the folder.
The programs in the folder become invisible.
No program in the folder can be output to an external device.
By setting the edit prohibition attribute or edit/display prohibition attribute, the programs in the folder can be protected from modification and deletion due to an operation error.

NOTE
Subfolders cannot be created in the MTB1 folder.

(4) Password and keyword
By setting a password in parameter No. 11311 and setting a keyword in parameter No. 11312, the attribute setting of the MTB1 folder can be locked.
In the locked state, the attribute of the MTB1 folder cannot be changed. Thus, the protection of the programs in the MTB1 folder cannot be canceled without setting the correct keyword.
- Locked state
The locked state means the state where the value of the password parameter No. 11311 and the value of the keyword parameter No. 11312 differ from each other.
The values set in these parameters are not displayed.
- Unlocked state
The unlocked state means the state where the value of the password parameter No. 11311 and the value of the keyword parameter No. 11312 match each other.
The values set in these parameters are not displayed.
- State where no key is set
When 0 is displayed in the password parameter No. 11311, no password is set yet.

NOTE
The attribute of the MTB1 folder can be changed only in the unlocked state.

(5) Calling
A program stored in the MTB1 folder can be used by calling based on a code such as M/T codes, macro calling based on a G code, macro calling based on G65/G66, and subprogram calling based on an M code such as M98.
For macro calling based on a G code, G codes used for calling are additionally set as indicated below.
Parameters are used to set the relationship between a G code used for macro calling and the number of a called program. Up to ten sets can be set.

Example)
Suppose that the following (Table 12.6.4 (a)) are set by the parameters:

Table 12.6.4 (a)

	First group	Second group	Third group
G code	No.12020=100	No.12023=150	No.12026=900
Program number	No.12021=8000	No.12024=7500	No.12027=8300
Number of G codes/programs	No.12022=10	No.12025=5	No.12028=30

Then the G codes call the programs in the Table 12.6.4 (b):

Table 12.6.4 (b)

G code	Called program
G100 to G109	O8000 to O8009
G150 to G154	O7500 to O7504
G900 to G929	O8300 to O8329

NOTE

When this function is enabled, the MTB1 folder is added to the top of the search order as a program search folder for macro call and subprogram call.

(1) If a program to be called is registered in the MTB1 folder, the program in the MTB1 folder is called.

(2) If a program to be called is not registered in the MTB1 folder, the program is searched for according to parameter No. 3457.

(6) Macro variable

Common variables #200 to #499 are added.

These variables are independent variables on a path-by-path basis as with variables #100 to #199.

- When the power is turned off, the variables are initialized and the data becomes null.
- When bit 3 (PV5) of parameter No. 6001 is set to 1, the custom macro outputs common variables #100 to #199, #200 to #499, and #500 to #599.
- When bit 6 (CCV) of parameter No. 6001 is set to 0, common variables #100 to #199 and #200 to #499 are cleared to null by a reset. When CCV is set to 1, these common variables are not cleared by a reset.

NOTE

By specifying parameter No. 6036, variables can be made common to all paths. If the option for the embedded macro function is specified but the addition of custom macro common variables is disabled (bit 6 (NCV) of parameter No.8135 is 1), #150 to #199 are unusable. However, in this parameter, set a number including #150 to #199.

Example of setting)

When 200 is set in parameter No. 6036:

#100 to #299: Common to all paths

#300 to #449: Used independently on a path-by-path basis

#150 to #199 can be used when the adding custom macro common variables is enabled (bit 6(NCV) of parameter No.8135 is 0).

Operation explanation

Procedure for creating programs in the MTB1 folder

(1) Creation

<1> Make the MTB1 folder empty.
Delete all programs and folders from the MTB1 folder.

<2> Create a program in the MTB1 folder.
A program can also be read through an external device.

<3> Set G code numbers for calling and program numbers to be called in parameters Nos. 12020 to 12049 as needed.

<4> Determine a password then specify it in parameter No. 11311.
Specify the same value as the password in parameter No. 11312 (keyword) to unlock the key.

<5> Set the desired attribute of the MTB1 folder.
<6> Set a value other than the password in parameter No. 11312 (keyword) to lock the key.

(2) Modification/addition
<1> Set the same value as the password in parameter No. 11312 (keyword) to unlock the key.
<2> Cancel the attribute of the MTB1 folder.
<3> Modify a program in the MTB1 folder or add a program to the MTB1 folder.
<4> Set the desired attribute of the MTB1 folder.
<5> Set a value other than the password in parameter No. 11312 (keyword) to lock the key.

(3) Copy (copy to another NC)
<1> Make the MTB1 folder empty.
Delete all programs and folders from the MTB1 folder.
<2> Read a program to the MTB1 folder through an external device.
<3> Set G code numbers for calling and program numbers to be called in parameters Nos. 12020 to 12049 as needed.
<4> Set a password in parameter No. 11311.
Set the same value as the password in parameter No. 11312 (keyword) to unlock the key.
<5> Set the desired attribute of the MTB1 folder.
<6> Set a value other than the password in parameter No. 11312 (keyword) to lock the key.

Setting of the attribute of the MTB1 folder

(1) Set the same value as the password in parameter No. 11312 (keyword) to unlock the key.
(2) Set the EDIT mode to display the program list screen.
Operate the soft keys in the order shown Fig. 12.6.4 (b).

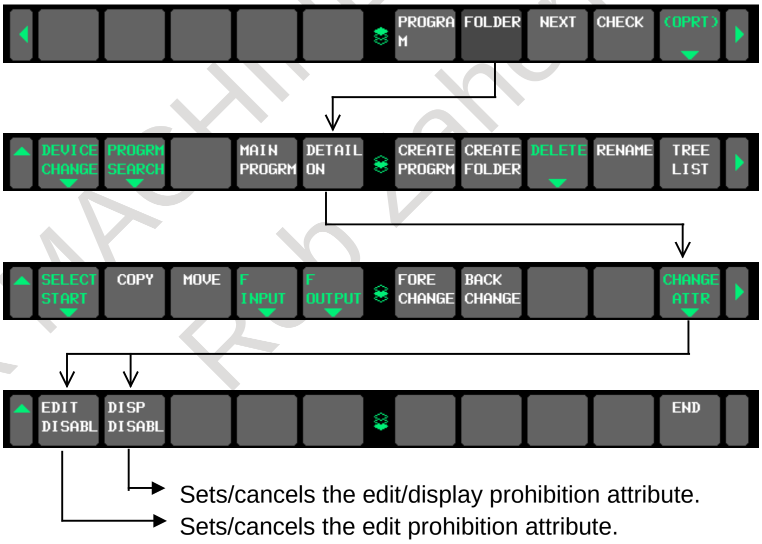


Fig. 12.6.4 (b)

(3) Place the cursor on the MTB1 folder then press a desired attribute soft key.

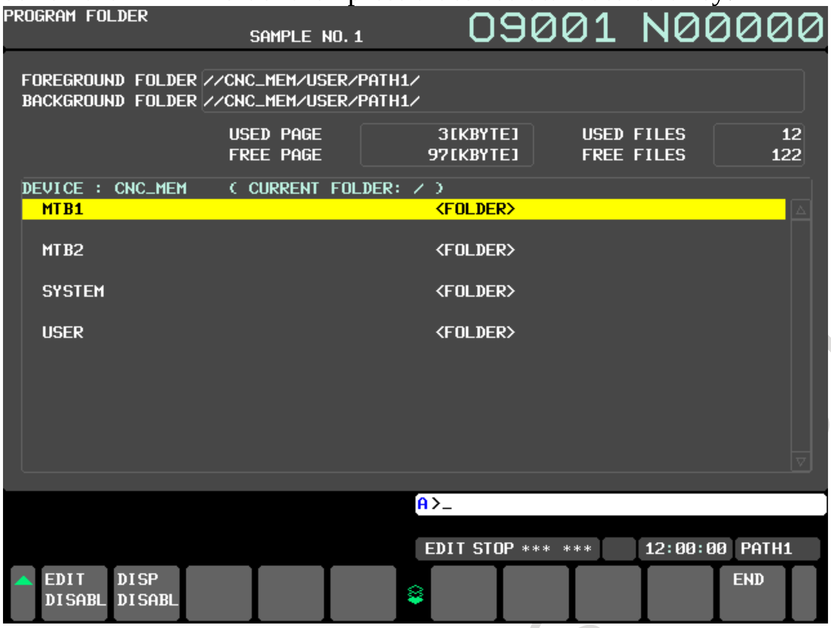


Fig. 12.6.4 (c) Attribute setting

Display when the edit prohibition attribute is assigned

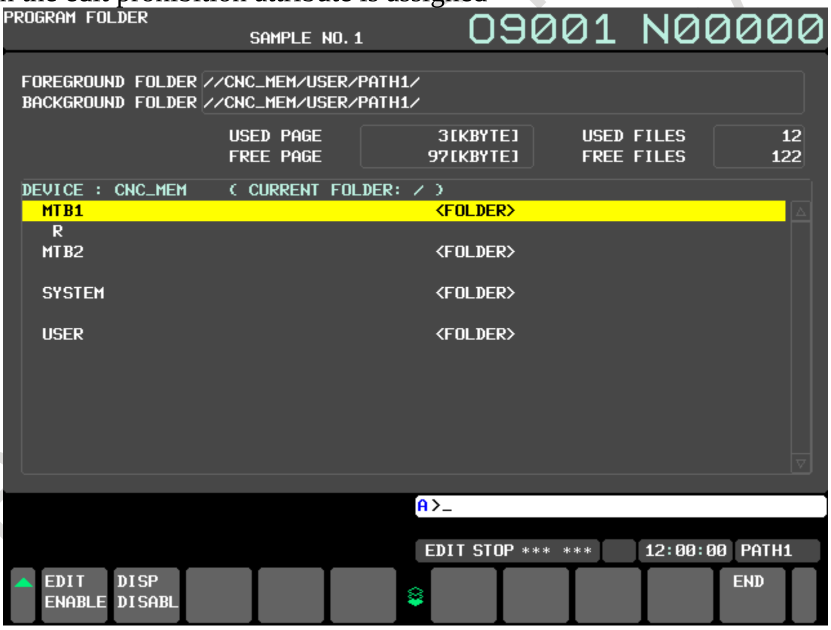


Fig. 12.6.4 (d) Edit prohibition attribute

Display when the edit/display prohibition attribute is assigned

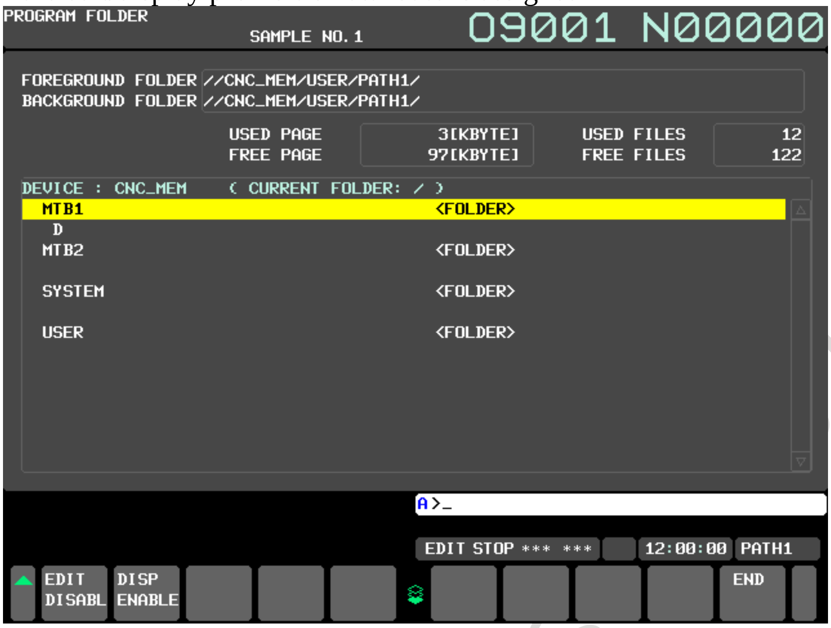


Fig. 12.6.4 (e) Edit/display prohibition attribute

(4) Set a value other than the password in parameter No. 11312 (keyword) to lock the key.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3457	SCF	SCC			SYS	MC1	MC2	LIB

[Input type] Parameter input
[Data type] Bit path

NOTE

1 The parameters LIB, MC2, MC1, and SYS are used to set a search folder for the following subprogram/macro calls:
- Subprogram call based on an M code
- Subprogram call based on a particular address
- Subprogram call based on a second auxiliary function code
- Macro call based on a G code
- Macro call based on an M code
- Macro call based on a T code
- One-touch macro call

2 The parameter SCF is used to set whether to add a search folder for the following subprogram/macro calls:
- Subprogram call based on M98
- Figure copy based on G72.1/G72.2
- Macro call based on G65/G66/G66.1
- Macro interrupt based on M96

#0 LIB The common program directory "//CNC_MEM/USER/LIBRARY/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

#1 MC2 MTB dedicated directory 2 "//CNC_MEM/MTB2/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

#2 MC1 MTB dedicated directory 1 "//CNC_MEM/MTB1/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

#3 SYS The system directory "//CNC_MEM/SYSTEM/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

NOTE
When multiple folders are specified, LIB, MC2, MC1, and SYS are searched in this order.

#6 SCC The same folder as the main program is added to the top of the search order as a search folder for the following each subprogram call and macro call.
- Subprogram call by an M code
- Subprogram call by a specific code
- Subprogram call by a second auxiliary function code
- Macro call by an S code
- Macro call by a T code
- Macro call by a G code
- Macro call by an M code
- One-touch macro call

The same folder as the main program is:
0: Not added in the search order.
1: Added in the search order.
When a search folder is added, a search is made in the following order:
0) Folder only for embedded macro (With the embedded macro-function.)
1) Folder where the main program is stored
2) Common program folder, which is an initial folder (LIBRARY)
3) MTB-dedicated folder 2, which is an initial folder (MTB2)
4) MTB-dedicated folder 1, which is an initial folder (MTB1)
5) System folder, which is an initial folder (SYSTEM)
The folders of 2) through 5) can be excluded from search target folders by setting the bits 0 (LIB), 1 (MC2), 2 (MC1), and 3 (SYS) of parameter No. 3457.

#7 SCF A search folder is:
0: Not added.
1: Added.

When a search folder is added, a search is made in the following order:
0) Folder only for embedded macro (With the embedded macro-function.)
1) Folder where the main program is stored
2) Common program folder, which is an initial folder (LIBRARY)
3) MTB-dedicated folder 2, which is an initial folder (MTB2)
4) MTB-dedicated folder 1, which is an initial folder (MTB1)
5) System folder, which is an initial folder (SYSTEM)

The folders of 3) through 5) can be excluded from search target folders by setting the bits 1 (MC2), 2 (MC1), and 3 (SYS) of parameter No. 3457.

	#7	#6	#5	#4	#3	#2	#1	#0
6001		CCV			PV5			

[Input type] Parameter input
[Data type] Bit path

#3 PV5 Custom macro common variables:
0: #500 to #549 are output. (Note)
1: #100 to #149 and #500 to 549 are output. (Note)

> **NOTE**
> Output variables are as the table according to the combination of added options.

When the parameter PV5=0

		Addition of custom macro common variables	
		Disable (bit 6 (NCV) of parameter No.8135 is 1)	Enable (bit 6 (NCV) of parameter No.8135 is 0)
Option “Embedded macro”	Non	#500 to #549	#500 to #999
	Yes	#500 to #549	#500 to #999

When the parameter PV5=1

		Addition of custom macro common variables	
		Disable (bit 6 (NCV) of parameter No.8135 is 1)	Enable (bit 6 (NCV) of parameter No.8135 is 0)
Option “Embedded macro”	Non	#100 to #149 and #500 to #549	#100 to #199 and #500 to #999
	Yes	#100 to #149, #200 to #499 and #500 to #549	#100 to #199, #200 to #499 and #500 to #999

#6 CCV Common variables #100 to #149(NOTE) cleared by power-off are:
0: Cleared to <null> by reset
1: Not cleared by reset

> **NOTE**
> Cleared variables are as the table according to the combination of added options.

		Addition of custom macro common variables	
		Disable (bit 6 (NCV) of parameter No.8135 is 1)	Enable (bit 6 (NCV) of parameter No.8135 is 0)
Option “Embedded macro”	Non	#100 to #149	#100 to #199
	Yes	#100 to #149 and #200 to #499	#100 to #199 and #200 to #499

6036	Number of custom macro variables common to tool path (for #100 to #199 (#499))

[Input type] Parameter input
[Data type] Word

[Valid data range] 0 to 400

When the memory common to paths is used, this parameter sets the number of custom macro common variables to be shared (custom macro variables common to paths). Common variables #100 to #199 (up to #499 in a system with the embedded macro option) may be shared. Ensure that the maximum number of usable macro common variables is not exceeded.

Example

When 20 is set in parameter No. 6036
#100 to #119: Shared by all paths
#120 to #149: Used by each path independently

NOTE

1 When the adding custom macro common variables is enabled (bit 6(NCV) of parameter No.8135 is 0), up to #199 is available.
2 To use up to #499, the embedded macro option is required.
3 When 0 or a negative value is set, the memory common to paths is not used.
4 When the option for embedded macro is effective and the addition of custom macro common variables is disabled (bit 6 (NCV) of parameter No.8135 is 1), #150 to #199 can not be used but this parameter should be set the number which includes #150 to #199.

	#7	#6	#5	#4	#3	#2	#1	#0
8106						CVP		

[Input type] Parameter input
[Data type] Bit

 # 2 CVP By a reset in path other than path 1, the custom macro variables common to paths set to the common variables #100 to #499 cleared by power-off is

0: not cleared to <null>.
1: cleared to <null>. (it depend to the bit 6(CCV) of parameter No.6001.)

NOTE

1 When a reset is made in path 1, it is cleared regardless of the setting of this parameter. (it depend to the parameter CCV)
2 This parameter is invalid for the path that the bit 0(NC1) of parameter No.6020 is 1.

11311	Password for embedded macro

[Input type] Parameter input
[Data type] 2-word
[Valid date range] 0 to 99999999

The password to set the attribute of the folder for the embedded macro (MTB1 folder) is set. When the values other than 0 are set to this parameter and the value is different from the parameter No. 11312 of the key word, the attribute of the MTB1 folder is locked.
Thereafter, the attribute of the MTB1 folder is locked unless the same value as the password is set to the key word. Moreover, the value of the password cannot be changed.

- When the key is open,
  The attribute of the MTB1 folder can be changed.

- When it locks or the key is not set,
  The attribute of the MTB1 folder can not be changed.

11312	Key word for embedded macro-function

[Input type] Parameter input
[Data type] 2-word
[Valid date range] 0 to 99999999

The key word in order to set the attribute of the folder for the embedded macro (MTB1 folder) is set.

> **NOTE**
> The value is not displayed even if the parameter is set. Moreover, when the power is turned off, this parameter becomes 0.

12020	G code number for the embedded macro (first)
12023	G code number for the embedded macro (second)
12026	G code number for the embedded macro (third)
12029	G code number for the embedded macro (fourth)
12032	G code number for the embedded macro (fifth)
12035	G code number for the embedded macro (sixth)
12038	G code number for the embedded macro (seventh)
12041	G code number for the embedded macro (eighth)
12044	G code number for the embedded macro (ninth)
12047	G code number for the embedded macro (tenth)

[Input type] Parameter input
[Data type] Word path
[Valid date range] 1 to 999

12021	Macro program number for the embedded macro (first)
12024	Macro program number for the embedded macro (second)
12027	Macro program number for the embedded macro (third)
12030	Macro program number for the embedded macro (fourth)
12033	Macro program number for the embedded macro (fifth)
12036	Macro program number for the embedded macro (sixth)
12039	Macro program number for the embedded macro (seventh)

12042	Macro program number for the embedded macro (eighth)

12045	Macro program number for the embedded macro (ninth)

12048	Macro program number for the embedded macro (tenth)

[Input type] Parameter input
[Data type] 2-word path
[Valid date range] 1 to 9999

12022	Number of G code macro for embedded macro (first)

12025	Number of G code macro for embedded macro (second)

12028	Number of G code macro for embedded macro (third)

12031	Number of G code macro for embedded macro (fourth)

12034	Number of G code macro for embedded macro (fifth)

12037	Number of G code macro for embedded macro (sixth)

12040	Number of G code macro for embedded macro (seventh)

12043	Number of G code macro for embedded macro (eighth)

12046	Number of G code macro for embedded macro (ninth)

12049	Number of G code macro for embedded macro (tenth)

[Input type] Parameter input
[Data type] Word path
[Valid date range] 1 to 255

The data of the macro call by G code added by the embedded macro is set. G code number and the macro program number for it are set, and the number of G codes is set. These sets can be set up to ten. If G code number duplicates, it gives priority from former set. The set whose G code number, macro program number, or number is 0 is invalid.

[Example] In case that the range of macro program number is 7000 to 8999:

	First group	Second group	Third group
G code	No. 12020=100	No. 12023=150	No. 12026=900
Program number	No. 12021=8000	No. 12024=7500	No. 12027=8300
Number	No. 12022=10	No. 12025=5	No. 12028=30

The following program is called by each G code.

G code	Called program
G100 toG109	O8000 to O8009
G150 to G154	O7500 to O7504
G900 to G929	O8300 to O8329

> **NOTE**
> The parameter value is regarded as 0 when data outside the range is set to the setting value of each parameter..

Note

> **NOTE**
> 1 The programs in the folder dedicated to embedded macros (MTB1 folder) can also be deleted by the following operations:
> - SRAM initialization operation
> - Clearing of program files with the IPL function
>
> 2 The programs in the MTB1 folder can be saved/restored in a batch by the following operations:
> - Saving/restoring of SRAM data in a batch with the BOOT function
> - Saving/restoring with the automatic data backup function (when programs are included as targets)
> - Output to an external device/input from an external device
>
> 3 To the programs in the MTB1 folder, the following protection functions are applied in addition to the protection of this function based on the folder attribute:
> - Parameter-based protection function for programs with O numbers in the eight thousands and nine thousands
> - Protection function based on a key and program encryption
> - Attribute setting for programs
>
> 4 This function differs from the embedded macro function of the FS16/18/21*i*-B in the following:
> - Program storage method
> - Program creation procedure
> - Program protection mechanism, etc.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	DATA INPUT/OUTPUT
	EDITING PROGRAMS
	PROGRAM MANAGEMENT

12.7 CANNED CYCLE FOR DRILLING

12.7.1 Canned Cycle for Drilling

Overview

Canned cycles for drilling make it easier for the programmer to create programs. With a canned cycle for drilling, a frequently-used machining operation can be specified in a single block with a G code; without canned cycles for drilling, normally more than one block is required. In addition, the use of canned cycles for drilling can shorten the program to save memory.

Explanation

A canned cycle for drilling consists of a sequence of six operations.

Operation 1 Positioning a hole position
Operation 2 Rapid traverse up to point R level
Operation 3 Hole machining

Operation 4 Operation at the bottom of a hole
Operation 5 Retraction to point R level
Operation 6 Rapid traverse up to the initial point

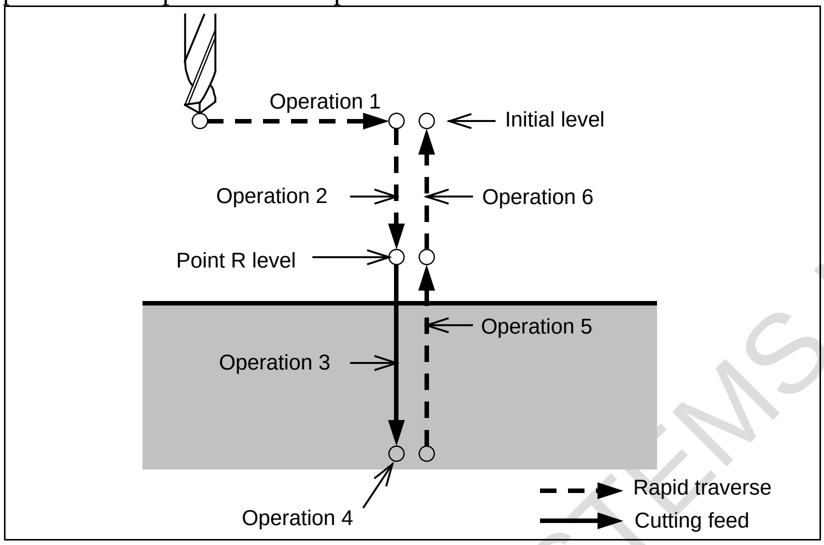


Fig. 12.7.1 (a) Canned cycle for drilling operation sequence

- Spindle control

In some canned cycles for drilling, a spindle command to rotate the spindle in reverse direction may be output automatically.

The following canned cycles for drilling require spindle control:

<Machining center system>		<Lathe system>	
Counter tapping cycle	G74	Face tapping cycle	G84
Fine boring cycle	G76	Side tapping cycle	G88
Tapping cycle	G84		
Boring cycle	G86		
Back boring cycle	G87		
Boring cycle	G88		

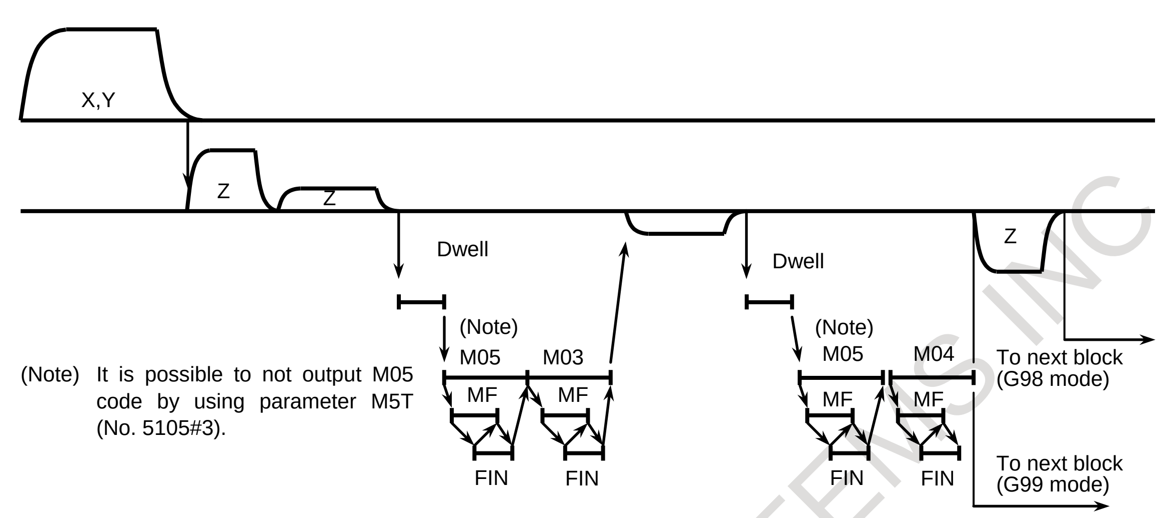
For spindle control, the following normal miscellaneous functions are used:
See the description of the miscellaneous functions.

M03 CW spindle rotation
M04 CCW spindle rotation
M05 Spindle stop
M19 Spindle orientation (machining center system)

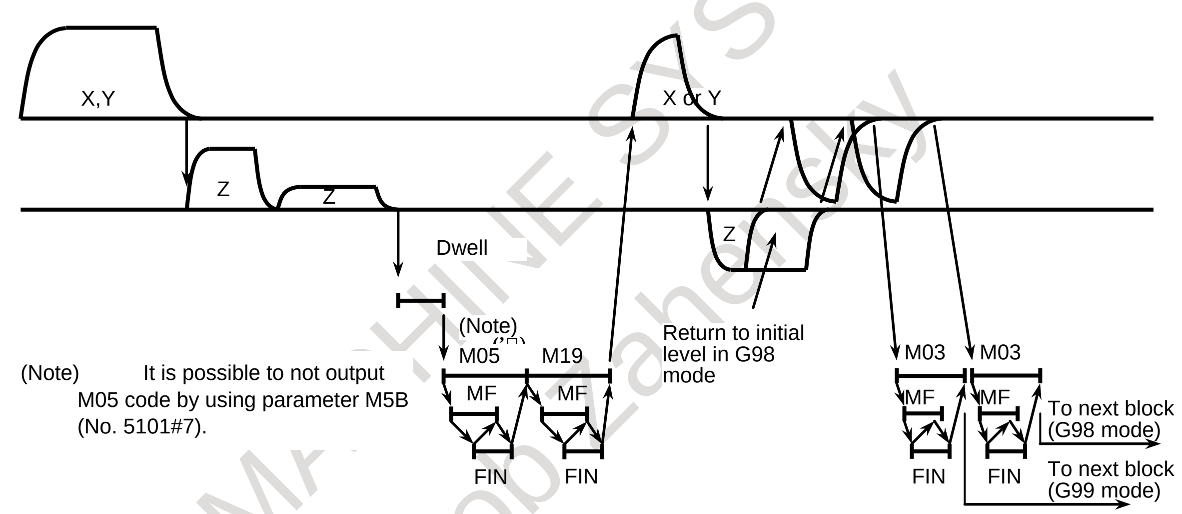
When the rotation direction of the spindle is to be switched from one direction to the other (for example, when M04 is output during M03 operation), a parameter can specify whether to send M05 at the time switching.

Timing charts are described in the following page:

G74 (Counter tapping cycle)



G76 (Fine boring cycle)



G84 (Tapping cycle)

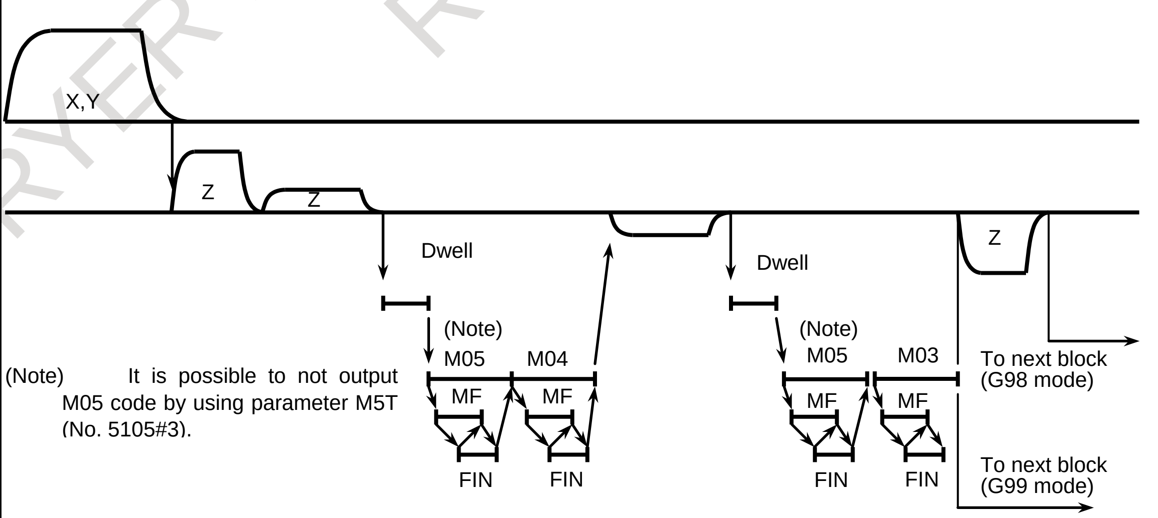


Fig. 12.7.1 (b) Machining of canned cycle for machining center system (1/2)

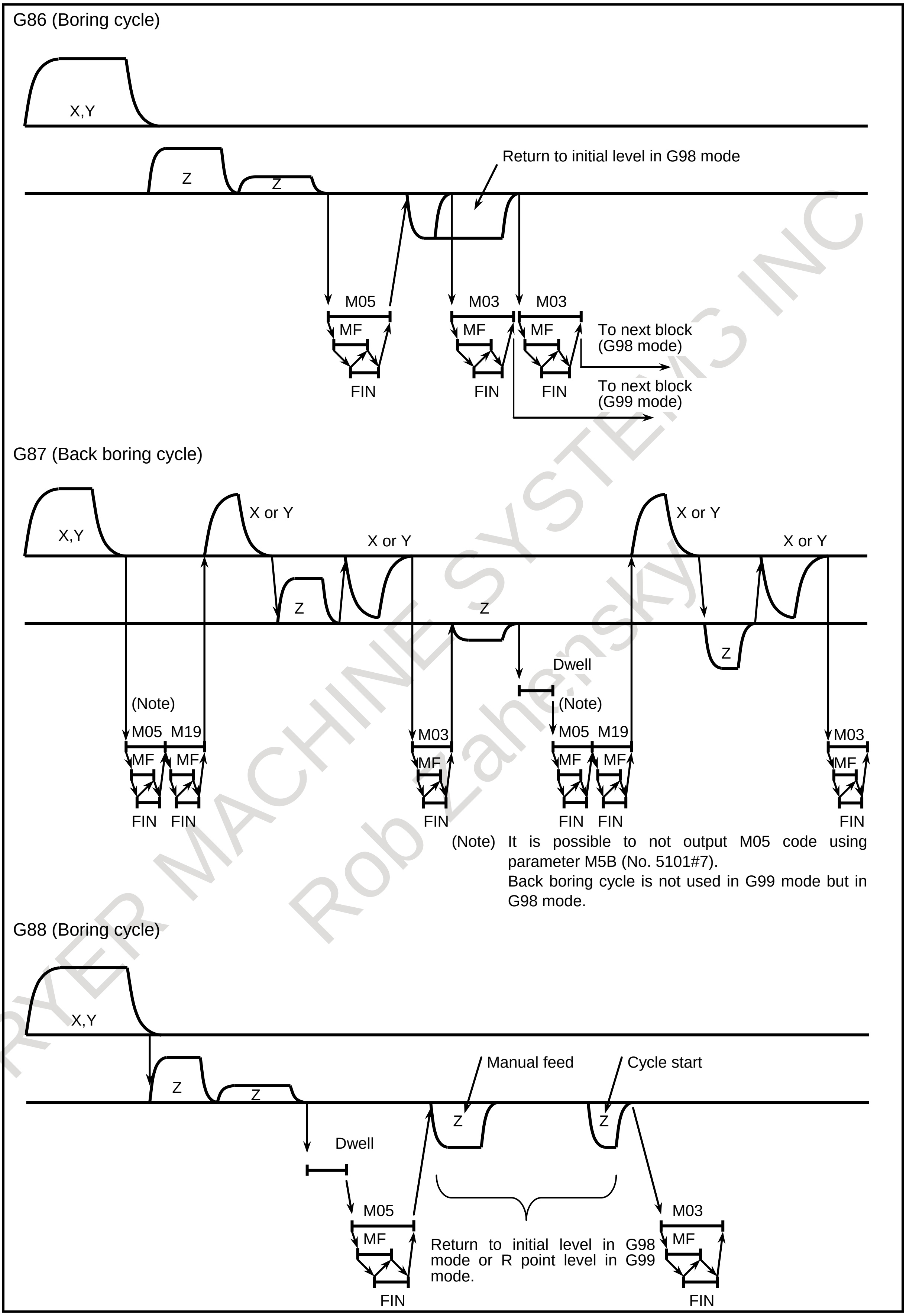


Fig. 12.7.1 (c) Machining of canned cycle for machining center system (2/2)

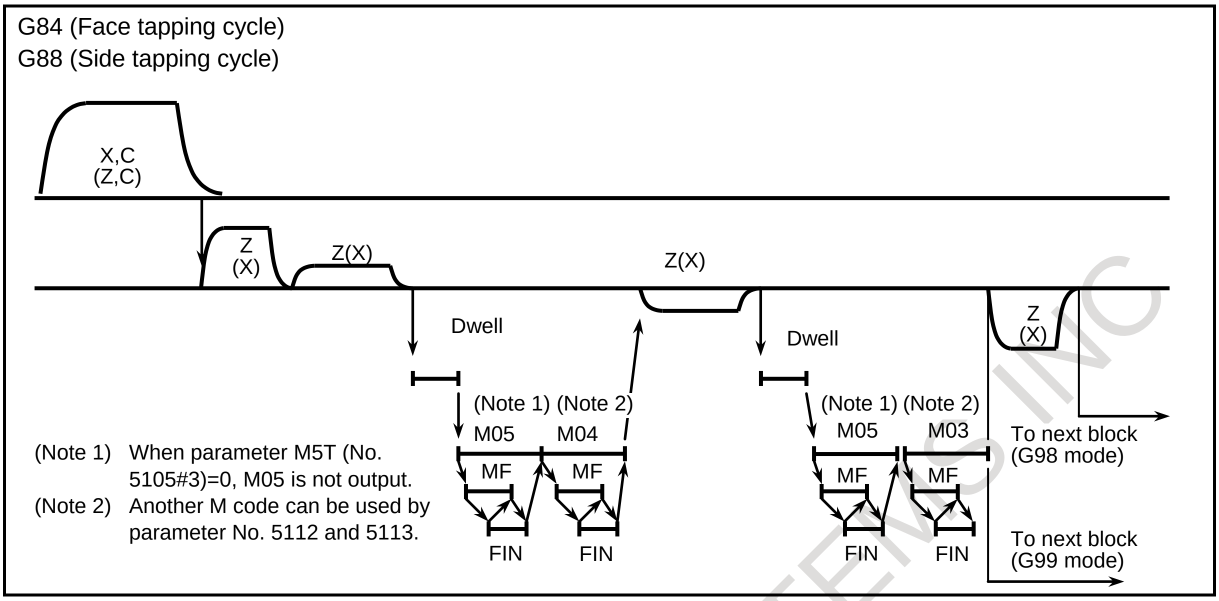


Fig. 12.7.1 (d) Machining of canned cycle for lathe system

- M code used for C-axis clamp/unclamp

T

When an M code specified in parameter No. 5110 for C-axis clamp is commanded, the CNC issues the M code for C-axis clamp after the tool is positioned and before the tool is fed in rapid traverse to the point-R level. The CNC also issues the M code (M code C-axis clamp +1) for C-axis unclamp after the tool retracts to the point-R level. The tool dwells for the time specified in parameter No. 5111.

- Tapping signal

During a tapping cycle, the tapping signal TAP <Fn001.5> is output. The tapping signal is output not only during the tapping machining but also during the G code of the tapping cycle is valid.

- Override

During tapping, cutting feedrate override is always set to 100%.

- Feed hold

When the feed hold key is pressed during tapping, the movement is not stopped immediately but the movement is stopped when the tool is returned to level R.

- Dry run

The bit 5 (TDR) of parameter No. 1401 specifies whether dry run is valid during tapping.

- Small-hole peck drilling cycle execution signal

M

In the small-hole peck drilling cycle mode, the small-hole peck drilling cycle execution signal PECK2 <Fn066.5> is output when positioning at point R on the drilling direction axis is started after positioning at a specified hole position is performed with G83 specified. This signal is not output if another canned cycle is specified or this mode is canceled by G80, a reset, or an emergency stop.

- Overload torque detection signal

M

For the overload torque detection signal in the small-hole peck drilling cycle, a skip signal SKIP#1 <X004.7>, SKIP#2 <X013.7>, or SKIP#3 <X011.7> is used. The skip signal is valid (a retract operation is performed) only when the tool on the drilling direction axis is located between point R and point Z and the tool is moving forward or performing a cutting operation.

Signal

Tapping signal TAP<Fn001.5>

[Classification] Output signal

[Function] Reports that the system is in tapping mode.

[Output cond.] The signal is set to “1” when:

- The system is in tapping cycle mode.
  (G74,G84 : for M series)
  (G84,G88 : for T series)
- The system is in tapping mode.
  (G63 : for M series)

The signal is set to “0” when:

- The system is in neither tapping cycle mode nor tapping mode.
- A reset or emergency stop is specified.

Overload torque detection signal SKIP$^{\#1}$<X004.7>, SKIP$^{\#2}$<X013.7>, SKIP$^{\#3}$<X011.7>

M

[Classification] Input signal

[Function] Refracts the tool on which an overload torque is imposed.

[Operation] When this signal is set to “1”, the control unit performs the following operation:

- When an overload torque is imposed, the tool is retracted to point R then machining is repeated after changing the spindle speed and cutting feedrate.
- This signal is valid (a retract operation is performed) only when the tool on the drilling axis is located between point R and point Z and the tool is moving forward or performing a cutting operation.

NOTE
This signal is used also as a skip signal.

Small-hole peck drilling cycle execution signal PECK2<Fn066.5>

M

[Classification] Output signal

[Function] Posts whether the small-hole peck drilling cycle is being executed.

[Output cond.] This signal is set to “1” for the following period:

- Period from the start time of positioning at point R on the drilling direction axis after positioning at a hole position is performed with G83 specified in this cycle mode until another canned cycle, which may be G80, or a G code of group 01 is specified or until this mode is canceled by a reset or emergency stop.
- This signal is not set to “1” in a state other than the above.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X004	SKIP#1							
X011	SKIP#3							
X013	SKIP#2							
Fn001			TAP					
Fn066			PECK2					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0001							FCV	

[Input type] Setting input
[Data type] Bit path

#1 FCV Program format
0: FANUC Series 16 program format
1: FANUC Series 15 program format

NOTE
1 Programs created in the FANUC Series 15 program format can be used for operation on the following functions:
   1 Subprogram call M98
   2 Thread cutting G32 (T series)
   3 Canned cycle G90, G92, G94 (T series)
   4 Multiple repetitive canned cycle G71 to G76 (T series)
   5 Drilling canned cycle G83.1, G80 to G89 (T series) / G73, G74, G76, G80 to G89 (M series)
2 When the FANUC Series 15 program format is used for this CNC, some limits may add. Refer to the Operator’s Manual.

	#7	#6	#5	#4	#3	#2	#1	#0
1401			TDR					

[Input type] Parameter input
[Data type] Bit path

#5 TDR Dry run during threading or tapping (tapping cycle G74 or G84, rigid tapping)
0: Enabled
1: Disabled

	#7	#6	#5	#4	#3	#2	#1	#0
3708		TSO						

[Input type] Parameter input
[Data type] Bit path

#6 TSO During a threading or tapping cycle, the spindle override is:
0: Disabled (tied to 100%).
1: Enabled.

> **NOTE**
> During rigid tapping, the override is tied to 100%, irrespective of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5101						RTR	EXC	FXY
	M5B						EXC	FXY

[Input type] Parameter input
[Data type] Bit path

#0 FXY The drilling axis in the drilling canned cycle, or cutting axis in the grinding canned cycle is:
0: Z-axis at all times.
1: Axis selected by the program

> **NOTE**
> In the case of the T series, this parameter is valid only for the drilling canned cycle in the FANUC Series 15 program format.

#1 EXC G81
0: Specifies a drilling canned cycle
1: Specifies an external operation command

#2 RTR G83 and G87
0: Specify a high-speed peck drilling cycle
1: Specify a peck drilling cycle

#7 M5B In drilling canned cycles G76 and G87:
0: Outputs M05 before an oriented spindle stops
1: Not output M05 before an oriented spindle stops

	#7	#6	#5	#4	#3	#2	#1	#0
5102	RDI	RAB			F16			

[Input type] Parameter input
[Data type] Bit path

#3 F16 When the FANUC Series 15 program format is used (with bit 1 (FCV) of parameter No. 0001 set to 1), a canned drilling cycle is specified using :
0: FANUC Series 15 program format
1: FANUC Series 16 program format. However, the number of repetitions is specified using address L.

#6 RAB When a canned drilling cycle using the FANUC Series 15 program format is specified (with bit 1 (FCV) of parameter No. 0001 set to 1 and bit 3 (F16) of parameter No. 5102 set to 0), address R specifies:
0: Increment command.
1: Absolute command with G code system A. With G code system B or C, G90 and G91 are followed.

#7 RDI When a canned drilling cycle using the FANUC Series 15 program format is specified (with bit 1 (FCV) of parameter No. 0001 set to 1 and bit 3 (F16) of parameter No. 5102 set to 0), address R is based on:
0: Radius specification.
1: Diameter/radius specification of the drilling axis.

	#7	#6	#5	#4	#3	#2	#1	#0
5103		TCZ			PNA	DCY		
		TCZ				DCY		SIJ

[Input type] Parameter input
[Data type] Bit path

#0 SIJ When the FANUC Series 15 program format is used (with bit 1 (FCV) of parameter No. 0001 set to 1), a tool shift value for the drilling canned cycle G76 or G87 is specified by:
0: Address Q. Set a tool retraction direction in parameter No. 5148.
1: Address I, J, or K.

#2 DCY When an axis (axis different from the drilling axis) perpendicular to the positioning plane is specified in a drilling canned cycle:
0: The specified axis is used as a drilling axis.
1: The axis specified in the block where the G code for the drilling canned cycle is specified is used as a drilling axis. The specified axis is used as a positioning axis.

NOTE
This parameter is valid when bit 0 (FXY) of parameter No. 5101 is set to 1.

#3 PNA In a drilling canned cycle using the FANUC Series 15 program format (with bit 1 (FCV) of parameter No. 0001 set to 1 and bit 3 (F16) of parameter No. 5102 set to 0), when a plane where no axis is present is specified in the drilling canned cycle mode:
0: An alarm is issued.
1: No alarm is issued.

#6 TCZ In a tapping cycle (excluding rigid tapping), an accumulated zero check in the tapping step (forward, backward) is:
0: Not performed.
1: Performed.

Execute a tapping cycle (excluding rigid tapping) with the servo feed forward (bit 1 (FEED) of parameter No. 2005). If an impact is detected, set this parameter to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
5105				K0D	M5T			SBC
					M5T			SBC

[Input type] Parameter input

[Data type] Bit path

#0 SBC In a drilling canned cycle, chamfer cycle, or corner rounding cycle:
- 0: A single block stop is not performed.
- 1: A single block stop is performed.

#3 M5T When the rotation direction of the spindle is changed from forward rotation to reverse rotation or from reserve rotation to forward rotation in a tapping cycle (G84/G88 with the T series, or G84/G74 with the M series):
- 0: M05 is output before output of M04 or M03.
- 1: M05 is not output before output of M04 or M03.

#4 K0D When K0 is specified in a drilling canned cycle (G80 to G89):
- 0: Drilling operation is not performed, but drilling data only is stored.
- 1: One drilling operation is performed.

5110	M code for C-axis clamping in canned cycles for drilling

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999998

This parameter sets the M code for C-axis clamping in canned cycles for drilling.

NOTE
When bit 4 (CME) of parameter No. 5161 is 1, the M code for C-axis clamping for the first pair is assumed.

5111	Dwell time when C-axis unclamping is specified in drilling canned cycle

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 32767
[Unit of data]

Increment system	IS-A	IS-B	IS-C	Unit
	10	1	0.1	msec

(The increment system does not depend on whether inch input or metric input is used.)

This parameter sets the dwell time when C-axis unclamping is specified in a drilling canned cycle.

5112	Spindle forward-rotation M code in drilling canned cycle

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999999

This parameter sets the spindle forward-rotation M code in a drilling canned cycle.

NOTE
M03 is output when 0 is set.

5113	Spindle reverse-rotation M code in drilling canned cycle

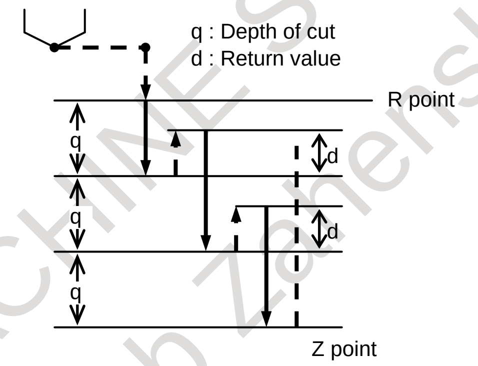
[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999999
This parameter sets the spindle reverse-rotation M code in a drilling canned cycle.

> **NOTE**
> M04 is output when 0 is set.

5114	Return value of high-speed peck drilling cycle

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the return value in high-speed peck drilling cycle.

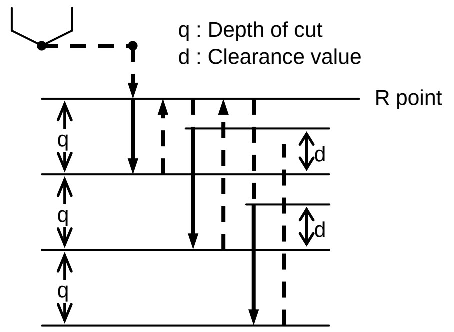
G73 (M series)
G83 (T series, when the bit 2 (RTR) of parameter No. 5101 is set to 0)



5115	Clearance value in a peck drilling cycle

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets a clearance value in a peck drilling cycle.

G83 (M series)
G83 (T series, when the bit 2 (RTR) of parameter No. 5101 is set to 1)



5148	Tool retraction direction after orientation in a fine boring cycle or back boring cycle

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] -24 to 24

This parameter sets an axis and direction for tool retraction after spindle orientation in a fine boring cycle or back boring cycle. For each boring axis, an axis and direction for tool retraction after orientation can be set. Set a signed axis number.

[Example] Suppose that:
When the boring axis is the X-axis, the tool retraction direction after orientation is -Y.
When the boring axis is the Y-axis, the tool retraction direction after orientation is +Z.
When the boring axis is the Z-axis, the tool retraction direction after orientation is -X.
Then, set the following (assuming that the first, second, and third axes are the X-axis, Y-axis, and Z-axis, respectively):
Set -2 in the parameter for the first axis. (The tool retraction direction is -Y.)
Set 3 in the parameter for the second axis. (The tool retraction direction is +Z.)
Set -1 in the parameter for the third axis. (The tool retraction direction is -X.)
Set 0 for other axes.

5149	Override for retraction in a boring cycle (G85/G89)

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 0 to 2000

This parameter sets an override value for the feedrate of retraction in a boring cycle. The cutting feedrate override signal and the second feedrate override signal are valid, regardless of the setting of this parameter. The setting of this parameter is valid even when the override cancel signal is set to “1”.
When 0 is set in this parameter, the following operation is performed:
For the T series
Operation performed when 200 is set in this parameter (The retraction feedrate is two times greater than the cutting feedrate.)
For the M series
Operation performed when 100 is set in this parameter (The retraction feedrate is the cutting feedrate.)

	#7	#6	#5	#4	#3	#2	#1	#0
5160		DNC			CYM			
				TSG	CYM	NOL	OLS	

[Input type] Parameter input
[Data type] Bit path

#1 OLS When an overload torque detection signal is received in a small-hole peck drilling cycle, the feedrate and spindle speed are:
0: Not changed.
1: Changed.

#2 NOL When the depth of cut per action is satisfied although no overload torque detection signal is received in a small-hole peck drilling cycle, the feedrate and spindle speed are:
0: Not changed.
1: Changed.

#3 CYM When a subprogram call is specified in a block specifying other commands in the canned cycle mode:
0: No alarm is issued. (When a command of address P is specified, the command is handled as both a command specifying a dwell time and a command specifying a subprogram number in a canned cycle.)
1: An alarm is issued.

#4 TSG The overload torque detection signal for a small-hole peck drilling cycle:
0: Depends on the parameter settings for the skip function.
1: Does not depend on the parameter settings for the skip function.

When this parameter is set to 1, the X address can be used for the overload torque signal even with the skip signal setting disabled. Even when the overload torque detection signal does not depend on the skip function parameter settings, parameter No. 3012 and bit 1 (SK0) of parameter No. 6200 remain valid.

#6 DNC When drilling axis is not commanded in the canned cycle command block for lathe system:
0: An alarm PS0566 "DRILLING AXIS IS NOT COMMANDED" is not issued.
1: An alarm PS0566 "DRILLING AXIS IS NOT COMMANDED" is issued.

NOTE
- The alarm PS0566 isn’t issued when drilling axis isn’t commanded except for the canned cycle command block for lathe system.

	#7	#6	#5	#4	#3	#2	#1	#0
5162						RED		
								RCK

[Input type] Parameter input
[Data type] Bit

#0 RCK When the command which relates to the reference position return such as G28 is commanded during the canned cycle for drilling:
0: Alarm PS0044 is not detected, and execute the program command.
1: Alarm PS0044 is detected.

#2 RED When selecting the drilling axis by the plane selection command in the canned cycle for drilling of the lathe system, the rotation axis (the axis on which the parameter ROTx (No.1006 # 0) = 1 is set) is
0: the target of the drilling axis.
1: excluded from the target of the drilling axis.

> **NOTE**
> - When using cylindrical interpolation, parallel axes of the basic three axes may be set in the parameter (No. 1022) of the rotation axis. In this case, if the rotation axis is excluded from the target of the drilling axis, set this parameter.

5163	M code that specifies the small-hole peck drilling cycle mode

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 99999999
This parameter sets an M code that specifies the small-hole peck drilling cycle mode.

5164	Percentage of the spindle speed to be changed at the start of the next advancing after an overload torque detection signal is received

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 255
This parameter sets the percentage of the spindle speed to be changed at the start of the next advancing after the tool is retracted because the overload torque detection signal is received.
$S2 = S1 \times d1 \div 100$
S1: Spindle speed to be changed
S2: Spindle speed changed
Set d1 as a percentage.

> **NOTE**
> When 0 is set, the spindle speed is not changed.

5165	Percentage of the spindle speed to be changed at the start of the next advancing when no overload torque detection signal is received

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 255
This parameter sets the percentage of the spindle speed to be changed at the start of the next advancing after the tool is retracted without the overload torque detection signal received.
$S2 = S1 \times d2 \div 100$
S1: Spindle speed to be changed
S2: Spindle speed changed
Set d2 as a percentage.

NOTE
When 0 is set, the spindle speed is not changed.

5166	Percentage of the cutting feedrate to be changed at the start of the next cutting after an overload torque detection signal is received

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 255

This parameter sets the percentage of the cutting feedrate to be changed at the start of cutting after the tool is retracted and advances because the overload torque detection signal is received.

$F2 = F1 \times b1 \div 100$

F1: Cutting feedrate to be changed
F2: Cutting feedrate changed

Set b1 as a percentage.

NOTE
When 0 is set, the cutting feedrate is not changed.

5167	Percentage of the cutting feedrate to be changed at the start of the next cutting when no overload torque detection signal is received

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 255

This parameter sets the percentage of the cutting feedrate to be changed at the start of cutting after the tool is retracted and advances without the overload torque detection signal received.

$F2 = F1 \times b2 \div 100$

F1: Cutting feedrate to be changed
F2: Cutting feedrate changed

Set b2 as a percentage.

NOTE
When 0 is set, the cutting feedrate is not changed.

5168	Lower limit of the percentage of the cutting feedrate in a small-hole peck drilling cycle mode

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 1 to 255

This parameter sets the lower limit of the percentage of the cutting feedrate changed repeatedly to the specified cutting feedrate.

$FL = F \times b3 \div 100$

F: Specified cutting feedrate
FL: Changed cutting feedrate

Set b3 as a percentage.

5170	Number of the macro variable to which to output the total number of retractions during cutting

[Input type] Parameter input
[Data type] Word path
[Valid data range] 100 to 149

This parameter sets the number of the custom macro common variable to which to output the total number of times the tool is retracted during cutting. The total number cannot be output to common variables #500 to #599.

5171	Number of the macro variable to which to output the total number of retractions because of the reception of an overload torque detection signal

[Input type] Parameter input
[Data type] Word path
[Valid data range] 100 to 149

This parameter sets the number of the custom macro common variable to which to output the total number of times the tool is retracted after the overload torque detection signal is received during cutting. The total number cannot be output to common variables #500 to #599.

5172	Feedrate of retraction to point R when no address I is specified

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets the feedrate of retraction to point R when no address I is specified.

5173	Feedrate of advancing to the position just before the bottom of a hole when no address I is specified

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets the feedrate of advancing to the position just before the bottom of a previously machined hole when no address I is specified.

5174	Clearance in a small-hole peck drilling cycle

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the clearance in a small-hole peck drilling cycle.

Alarm and message

Number	Message	Description
PS0044	G27-G30 NOT ALLOWED IN FIXED CYC	The command which relates to the reference position return is commanded in canned cycle mode for drilling. Modify the program.
PS0045	ADDRESS Q NOT FOUND (G73/G83)	In a high-speed peck drilling cycle or peck drilling cycle, the amount of each-time cutting is not specified. Modify the program.
PS0566	DRILLING AXIS IS NOT COMMANDED	When the parameter DNC (No.5160#6) is set to 1, drilling axis is not commanded in the canned cycle command block for lathe system. Modify the program.
PS5329	M98 AND NC COMMAND IN SAME BLOCK	A subprogram call which is not a single block was commanded during canned cycle mode. Modify the program.

Diagnosis data

520	Total number of retract operations during cutting after G83 is specified

521	Total number of retract operations based on reception of the overload torque detection signal during cutting after G83 is specified

> **NOTE**
> The total number of retract operations output to Nos. 520 and 521 is cleared to zero by the G83 command after the small-hole peck drilling cycle mode is set.

522	Coordinate on the drilling axis where a retract operation was started (least input increment)

523	Difference between the coordinate on the drilling axis where the previous retract operation was started and the coordinate on the drilling axis where the current retract operation was started (least input increment: previous value - current value)

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Canned cycle for drilling

12.7.2 Canned Cycle Overlap for Drilling

Overview

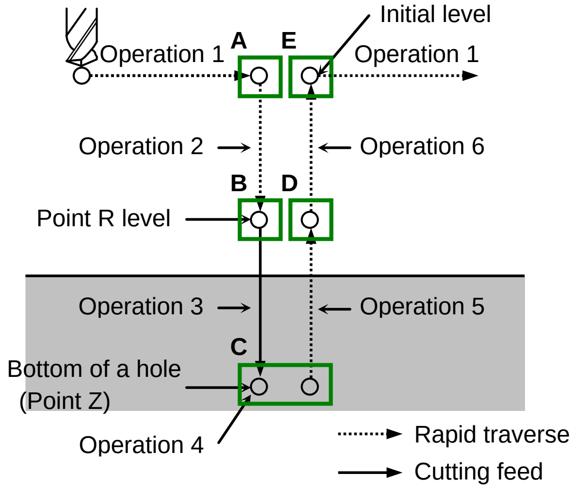
With this function, during the canned cycle mode for drilling, the command can overlap with the next block for the settime of the time constant of the acceleration/deceleration after interpolation.
This speeds up the operation of the canned cycle for drilling and shortens the cycle time.

Explanation

This function is valid when bit 0 (HDCx) of parameter No.1680 is set to 1.

- Operations in which overlap is valid

Fig. 12.7.2 (a) Operations in which overlap is shows the operations in which overlap is valid in the operation sequence of canned cycle for drilling.
When bits 0 to 5 of parameter No.1681 are set to 1, overlap is invalid in each operation.



Operation 1 : Positioning
Operation 2 : Rapid traverse up to point R level
Operation 3 : Hole machining
Operation 4 : Operation at the bottom of a hole
Operation 5 : Retraction to point R level
Operation 6 : Rapid traverse up to the initial point

Point A - Point E : Overlap is valid.

When canned cycle for drilling is canceled at point E, no overlap is done at point E.
Also, since the operation varies slightly depending on the type of cycle, please refer to "Canned Cycle Overlap for Drilling" of OPERATOR'S MANUAL (B-64694EN) for details.

Fig. 12.7.2 (a) Operations in which overlap is valid

NOTE
When overlap is done, in-position check is invalid.

- Supported canned cycle for drilling

Table 12.7.2 (a), Table 12.7.2 (b) Canned cycle for drilling (Lathe system), Table 12.7.2 (c) Canned cycle for drilling in series 15 program format (Lathe system) shows canned cycle for drilling supported by this function.

M

Table 12.7.2 (a) Canned cycle for drilling (Machining center system)

G code	Drilling	Operation at the bottom of a hole	Retraction	Application
G73	Intermittent feed	-	Rapid traverse	High-speed peck drilling cycle
G81	Cutting feed	-	Rapid traverse	Drilling cycle, spot drilling cycle
G82	Cutting feed	Dwell	Rapid traverse	Drilling cycle, counter boring cycle
G83	Intermittent feed	-	Rapid traverse	Peck drilling cycle
G85	Cutting feed	-	Cutting feed	Boring cycle
G86	Cutting feed	Spindle stop	Rapid traverse	Boring cycle
G89	Cutting feed	Dwell	Cutting feed	Boring cycle

T

Table 12.7.2 (b) Canned cycle for drilling (Lathe system)

G code	Drilling axis	Drilling	Operation at the bottom of a hole	Retraction	Application
G83	Z axis	Intermittent feed Cutting feed	Dwell	Rapid traverse	Front drilling cycle
G85	Z axis	Cutting feed	Dwell	Cutting feed	Front boring cycle
G87	X axis	Intermittent feed Cutting feed	Dwell	Rapid traverse	Side drilling cycle
G89	X axis	Cutting feed	Dwell	Cutting feed	Side boring cycle

Table 12.7.2 (c) Canned cycle for drilling in series 15 program format (Lathe system)

G code	Drilling	Operation at the bottom of a hole	Retraction	Application
G83.1	Intermittent feed	-	Rapid traverse	High-speed peck drilling cycle
G81	Cutting feed	-	Rapid traverse	Drilling cycle, spot drilling cycle
G82	Cutting feed	Dwell	Rapid traverse	Drilling cycle, counter boring cycle
G83	Intermittent feed	-	Rapid traverse	Peck drilling cycle
G85	Cutting feed	-	Cutting feed	Boring cycle
G89	Cutting feed	Dwell	Cutting feed	Boring cycle

- Overlap time

Set the overlap time of this function to the parameters No.1686 and No.1687.
Parameters No.1686 and No.1687 are used as the time constant of the acceleration/deceleration after interpolation common to both cutting feed and rapid traverse in canned cycle for drilling in which this function is valid.
The command can overlap with the next block for the overlap time set within the time constant of the acceleration/deceleration after interpolation. (Fig.12.7.2 (b))

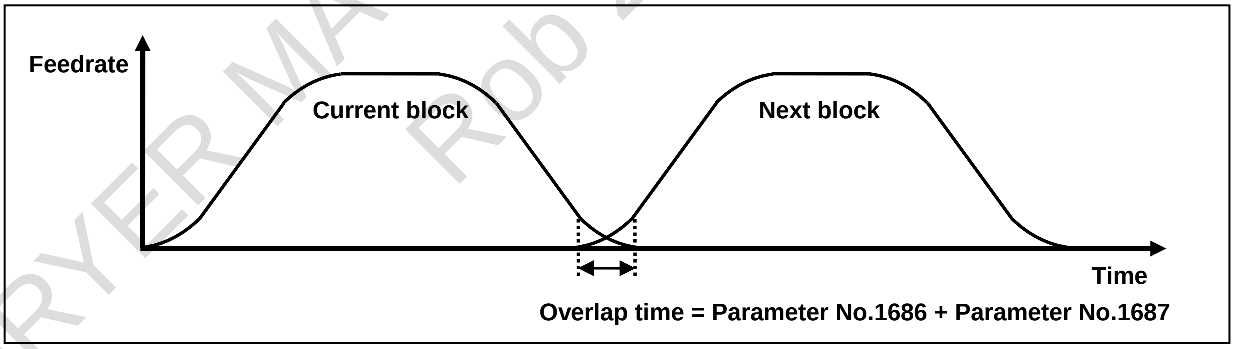


Fig.12.7.2 (b) Overlap time

Parameters No.11248 and No.11249 are valid for axes whose parameters No.1686 and No.1687 are both set to 0. (Table 12.7.2 (d))
Parameters No.11248 and No.11249 are the overlap time of Smart overlap. For Smart overlap, refer to "Smart Overlap" in this manual.

Table 12.7.2 (d) Relationship between parameter setting and time constant of acceleration/deceleration after interpolation for Canned cycle overlap for drilling

Parameter No.1686	Parameter No.1687	Time constant 1 of acceleration/deceleration in Canned cycle overlap for drilling	Time constant 2 of acceleration/deceleration in Canned cycle overlap for drilling
0	0	Parameter No.11248 is valid.	Parameter No.11249 is valid.
Other than 0	0	Parameter No.1686 is valid.	Parameter No.1687 is valid.
0	Other than 0	Parameter No.1686 is valid.	Parameter No.1687 is valid.
Other than 0	Other than 0	Parameter No.1686 is valid.	Parameter No.1687 is valid.

This function has the following features, so usually the overlap amount can be increased compared to Smart overlap.

- Since it is limited to operation within a cycle, interference due to overlap is less likely to be a problem.
- Overlap can be invalidated for each cycle operation.

To reduce the cycle time, it is recommended that the overlap time of this function is set to the parameters No.1686 and No.1687 separately from Smart overlap.

Normally, the overlap time constant 2 is not used. Set the parameters No.1687 and No.11249 to 0.

- Acceleration/deceleration after interpolation during canned cycle for drilling

The following parameters are used as the time constant of the acceleration/deceleration after interpolation in canned cycle for drilling in which this function is valid.
The relationship between the parameters and the time constant of the acceleration/deceleration is shown in Table 12.7.2 (e)and Fig. 12.7.2 (c).

Table 12.7.2 (e) Acceleration/deceleration after interpolation during canned cycle for drilling

Move command	Time constant of the acceleration/deceleration
Rapid traverse	T1 : Parameter No.1620 T2 : Parameter No.1682 T3 : Parameter No.1686(Overlap time)
Cutting feed	T1 : Parameter No.1683 T2 : None T3 : Parameter No.1687 (Overlap time)

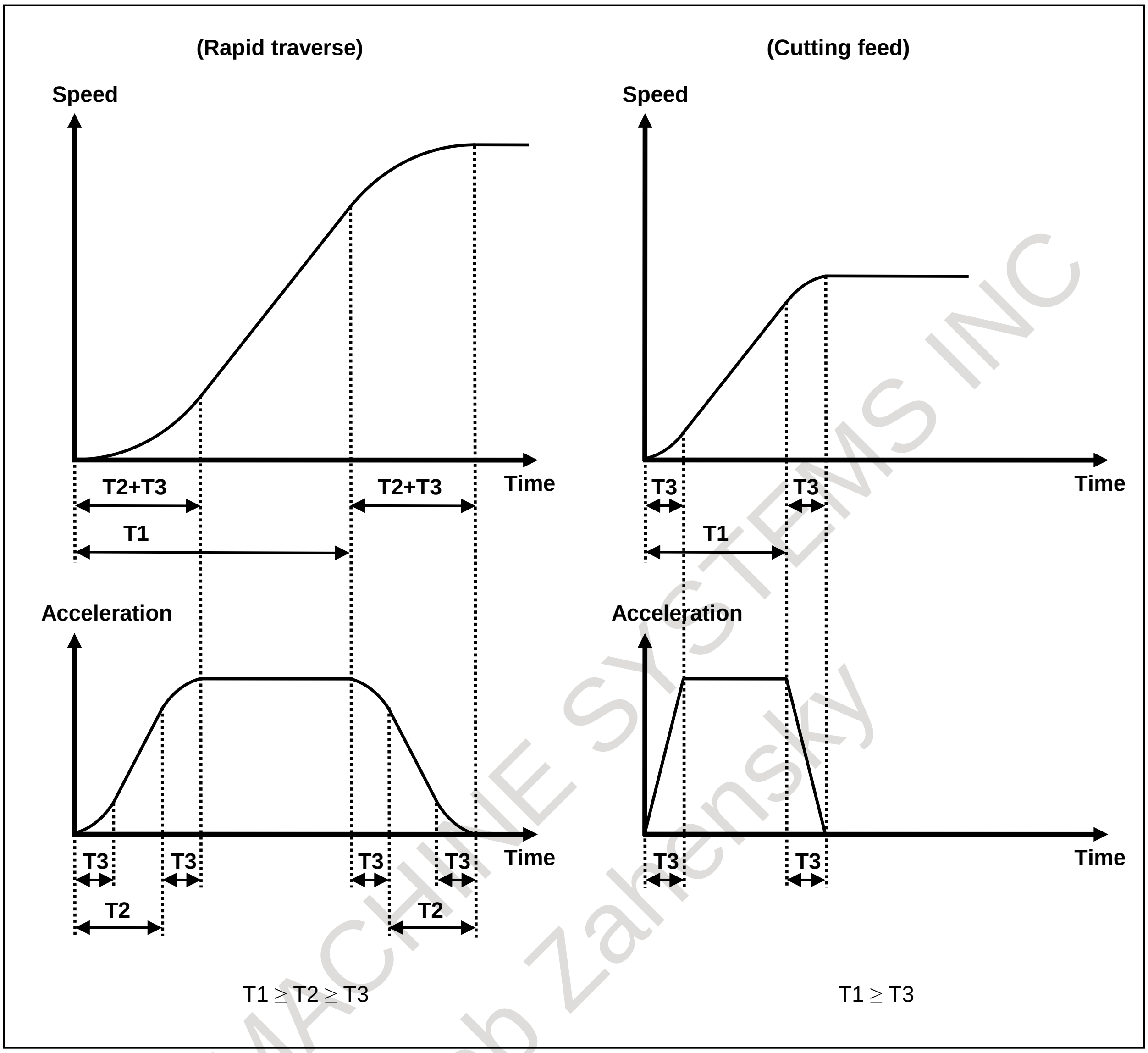


Fig. 12.7.2 (c) Acceleration/deceleration after interpolation during canned cycle for drilling

- Acceleration/deceleration during canned cycle for drilling (When the acceleration/deceleration before interpolation is valid)

The following parameters are used as the time constant of the acceleration/deceleration after interpolation in canned cycle for drilling in which this function is valid.

When the acceleration/deceleration before interpolation is valid, such as AI contour control I and AI contour control II, the relationship between the parameters and the time constant of the acceleration/deceleration is shown in Table 12.7.2 (f).

Table 12.7.2 (f) Acceleration/deceleration during canned cycle for drilling (When the acceleration/deceleration before interpolation is valid)

Move command	Time constant of the acceleration/deceleration
Rapid traverse	T1 : Parameter No.1671 (Acc/dec before interpolation) (Maximum allowable acceleration rate) T2 : Parameter No.1672 (Acc/dec before interpolation) Parameter No.1684 (Acc/dec after interpolation) T3 : Parameter No.1686 (Acc/dec after interpolation) (Overlap time)

Move command	Time constant of the acceleration/deceleration
Cutting feed	T1 : Parameter No.1660 (Acc/dec before interpolation) (Maximum permissible acceleration) T2 : Parameter No.1772 (Acc/dec before interpolation) Parameter No.1685 (Acc/dec after interpolation) T3 : Parameter No.1686 (Acc/dec after interpolation) (Overlap time)

- Tool path error at corner by overlap

When overlap is done where the movement direction changes, the overlapped path passes inward compared to the programmed path. (Fig. 12.7.2 (d))

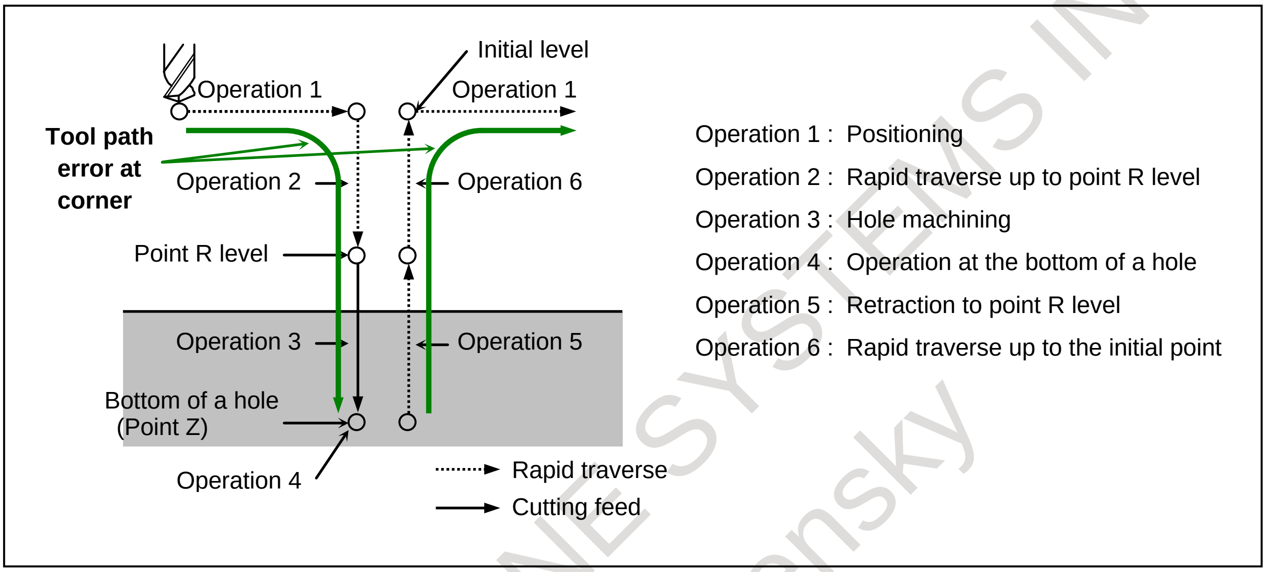


Fig. 12.7.2 (d) Tool path error at corner by overlap

When the overlap operations are continuous, overlap is performed not only between two blocks, but also over two or more blocks. (Fig. 12.7.2 (e))

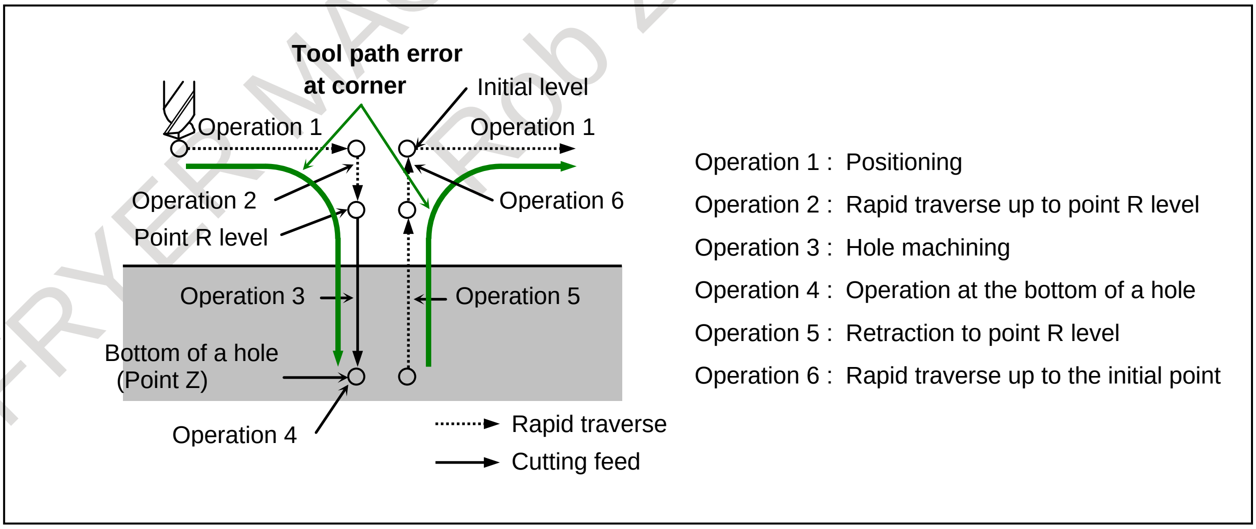


Fig. 12.7.2 (e) Tool path error at corner by overlap (2)

NOTE

It is necessary to care that tool does not interfere with workpiece by tool path error at corner.
When bit 0 (DPS) of parameter No.1681 is set to 1, overlap is invalid at corner. If it is required to avoid tool path error at corner, please set DPS to 1.

- Tool path error at bottom of a hole by overlap

When dwell is not commanded at the bottom of a hole, overlap is done at the bottom of the hole.
In this case, the tool does not reach the programmed hole bottom because the direction of movement is reversed before the programmed hole bottom. (Fig. 12.7.2 (f))

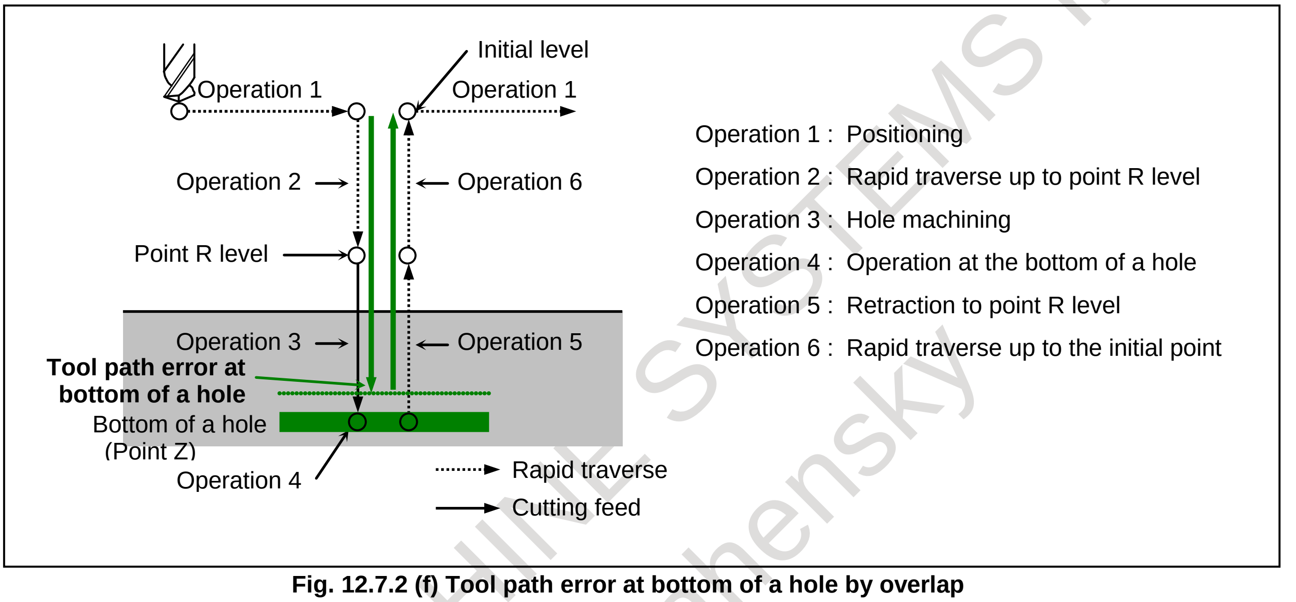


Fig. 12.7.2 (f) Tool path error at bottom of a hole by overlap

NOTE

When bit 5 (DZL) of parameter No.1681 is set to 1, overlap is invalid at bottom of a hole. If accuracy at bottom of a hole is required, please set DZL to 1.

- Relationships with other functions

The relationships with other functions are as follows.

- Rapid traverse block overlap

This function can be used with rapid traverse block overlap.
When these functions are used at the same time, overlap is done at the earlier timing of this function and rapid traverse block overlap.

- Cs contour control

When positioning is executed with the Cs contour control axis, the in-position check of the Cs contour control axis is done even when the overlap is valid.

- Limitation

This function cannot be used with the following functions.

- Small-hole peck drilling cycle (G83) (Machining center system)
- 3-dimensional coordinate system conversion
- Tilted working plane indexing (Machining center system)
- Angular axis control
- Retrace (Machining center system)
- Manual handle retrace

- Synchronous/Composite control (Lathe system)
- Superimposed control (Lathe system)
- Index table indexing (Machining center system)

Note

NOTE

1 When overlap is done between the rapid traverse block and the cutting feed block, the feedrate might be higher than the cutting feedrate at the start position of the cutting feed block. This is more likely to occur when the overlap time (parameter No.1686 and No.1687) is too large againstthe time constant of the acceleration/deceleration. In such a case, please adjust to reduce the overlap time.

2 When the feed-forward function is used at the same time, shock may occur between the rapid traverse block and the cutting feed block. In such a case, please use the following servo software and validate the cutting/rapid feed-forward switching function (bit 4 of parameter No.2214 is 1).

[Series and editions of applicable servo software]

- Series 90J0 / 04.0 and subsequent editions
- Series 90K0 / 04.0 and subsequent editions

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1680								HDCx

[Input type] Parameter input
[Data type] Bit axis

NOTE

When this parameter is set, the power must be turned off before operation is continued.

#0 HDCx Canned cycle overlap for drilling is :
0: Invalid.
1: valid.

	#7	#6	#5	#4	#3	#2	#1	#0
1681			DZL	DQL	DFW	DRV	DRL	DPS

[Input type] Parameter input
[Data type] Bit path

#0 DPS In canned cycle overlap for drilling, for "between the rapid traverse block to the hole position and the rapid traverse block to point R level" and "between the return block to point R level / initial level and the rapid traverse block to the next hole position", the overlap is:
0: Valid.
1: Invalid.

#1 DRL In canned cycle overlap for drilling, for "between the rapid traverse block to point R level and the hole machining block" and "between the retraction block to point R level and the return block to initial level", overlap is:

0: Valid.
1: Invalid.

#2 DRV In canned cycle overlap for drilling, for "between the retraction block and the hole machining block for (high-speed) peck drilling cycle", overlap is:
0: Valid.
1: Invalid.

#3 DFW In canned cycle overlap for drilling, for "between the hole machining (rapid traverse) block and the hole machining (cutting feed) block for peck drilling cycle", overlap is:
0: Valid.
1: Invalid.

#4 DQL In canned cycle overlap for drilling, for "between the hole machining block and the retraction block for (high-speed) peck drilling cycle", overlap is:
0: Valid.
1: Invalid.

#5 DZL In canned cycle overlap for drilling, for the bottom of a hole, overlap is:
0: Valid.
1: Invalid.

1682	Time constant T2 used for acceleration/deceleration of rapid traverse for each axis in canned cycle for drilling

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 512

In canned cycle overlap for drilling, set the time constant T2 used for acceleration/deceleration of rapid traverse for each axis in canned cycle for drilling.

1683	Time constant T1 used for acceleration/deceleration of cutting feed for each axis in canned cycle for drilling

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

In canned cycle overlap for drilling, set the time constant T1 used for acceleration/deceleration of cutting feed for each axis in canned cycle for drilling.

1684	Time constant used for acceleration/deceleration of rapid traverse for each axis in canned cycle for drilling (Time constant used for acceleration/deceleration after interpolation of rapid traverse in acceleration/deceleration before interpolation mode)

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

In canned cycle overlap for drilling, set the time constant used for acceleration/deceleration after interpolation of rapid traverse for each axis in acceleration/deceleration before interpolation mode (AI contour control I and AI contour control II) and canned cycle for drilling.

1685	**Time constant used for acceleration/deceleration of cutting feed for each axis in canned cycle for drilling (Time constant used for acceleration/deceleration after interpolation of cutting feed in acceleration/deceleration before interpolation mode)**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

In canned cycle overlap for drilling, set the time constant used for acceleration/deceleration after interpolation of cutting feed for each axis in acceleration/deceleration before interpolation mode (AI contour control I and AI contour control II) and canned cycle for drilling.

1686	**Time constant 1 of acceleration/deceleration after interpolation in Canned cycle overlap for drilling for each axis**

[Input type] Parameter input
[Data type] word axis
[Unit of data] msec
[Valid data range] 0 to 64

Set the time constant 1 of acceleration/deceleration after interpolation in Canned cycle overlap for drilling for each axis.

> **NOTE**
> When both parameter No.1686 and No.1687 are set to 0, parameters No.11248 and No.11249 are valid for the time constant of acceleration/deceleration after interpolation in Canned cycle overlap for drilling.

1687	**Time constant 2 of acceleration/deceleration after interpolation in Canned cycle overlap for drilling for each axis**

[Input type] Parameter input
[Data type] word axis
[Unit of data] msec
[Valid data range] 0 to 64

Set the time constant 2 of acceleration/deceleration after interpolation in Canned cycle overlap for drilling for each axis.

> **NOTE**
> When both parameter No.1686 and No.1687 are set to 0, parameters No.11248 and No.11249 are valid for the time constant of acceleration/deceleration after interpolation in Canned cycle overlap for drilling.

11248	**Time constant 1 of acceleration/deceleration after interpolation in Smart overlap / Canned cycle overlap for drilling for each axis**

[Input type] Parameter input
[Data type] word axis
[Unit of data] msec
[Valid data range] 0 to 64

Set the time constant 1 of acceleration/deceleration after interpolation in Smart overlap / Canned cycle overlap for drilling for each axis.

> **NOTE**
> When either parameter No.1686 or No.1687 is set to other than 0, parameters No.1686 and No.1687 are valid for the time constant of acceleration/deceleration after interpolation in Canned cycle overlap for drilling.

11249	Time constant 2 of acceleration/deceleration after interpolation in Smart overlap / Canned cycle overlap for drilling for each axis

[Input type] Parameter input
[Data type] word axis
[Unit of data] msec
[Valid data range] 0 to 64

Set the time constant 2 of acceleration/deceleration after interpolation in Smart overlap / Canned cycle overlap for drilling for each axis.

> **NOTE**
> When either parameter No.1686 or No.1687 is set to other than 0, parameters No.1686 and No.1687 are valid for the time constant of acceleration/deceleration after interpolation in Canned cycle overlap for drilling.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Canned cycle overlap for drilling

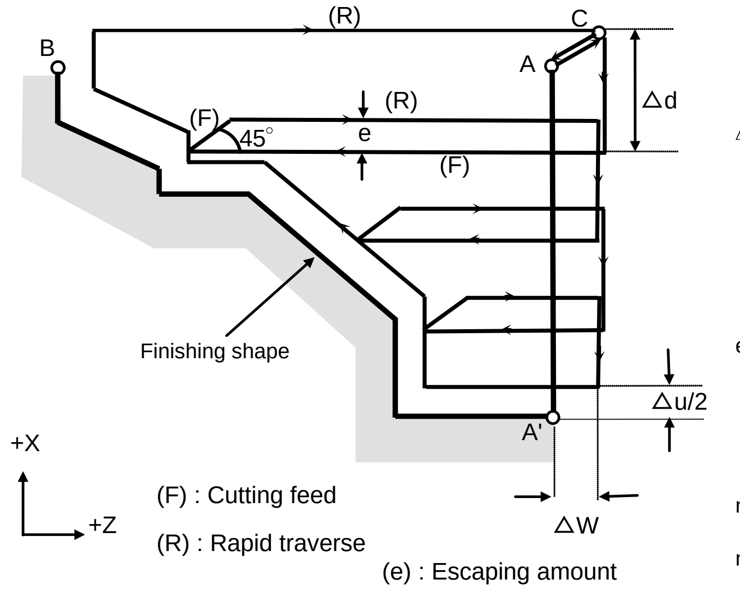
12.8 CANNED CYCLE / MULTIPLE REPETITIVE CANNED CYCLE

12.8.1 CANNED CYCLE / MULTIPLE REPETITIVE CANNED CYCLE

T

Overview

This function provides several predetermined canned cycles to make CNC programming easy. For instance, the intermediate tool path for rough machining is determined automatically by specifying the data of only the finish work shape. And also, a canned cycle for the thread cutting is available.
The following example shows stock removals in turning type I.
If a finished shape of A→A'→B is given by a program as in the figure below, the specified area is removed by Δd (depth of cut), with finishing allowance Δu/2 and Δw left.



G71 U(Δd) R(e);
G71 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t);
N(ns). . . .
. . . .
F_
S_
T_
N(nf) . . . . ;

The move command of a finished shape of A→A'→B is specified in the blocks from sequence number ns to nf.

- Δd : Depth of cut (radius designation)
  Designate without sign. The cutting direction depends on the direction AA'. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5132), and the parameter is changed by the program command.
- e : Escaping amount
  This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5133), and the parameter is changed by the program command.
- ns : Sequence number of the first block for the program of finishing shape.
- nf : Sequence number of the last block for the program of finishing shape.
- Δu : Distance and direction of finishing allowance in X direction (diameter / radius designation).
- Δw : Distance and direction of finishing allowance in Z direction.
- f,s,t : Any F , S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G71 block is effective.

Signal

Chamfering signal *CDZ<Gn053.7>

[Classification] Input signal

[Function] Executes chamfering in a threading cycle (G92 (T series), or G76 (T series)). Specify the chamfering distance in parameter No. 5130.

[Operation] When the signal is set to 1, chamfering is not executed in the threading cycle.
When the signal is set to 0, chamfering is executed in the threading cycle.

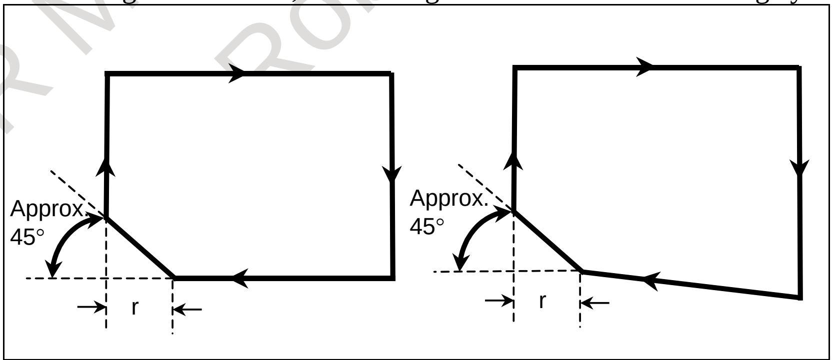


Fig. 12.8.1 (a) Straight threading and taper threading

Set the chamfering distance r to the parameter No. 5130. When the multiple repetitive canned cycle is enabled (bit 3 (NMR) of parameter No.8137 is 0), the chamfering distance can be specified in G76 or G76.7.

The chamfering angle is made smaller than 45° by the remaining pulses in the automatic acceleration/ deceleration circuit and servo system. In addition, the angle can be changed to other than 45° by setting an angle for parameter No. 5131.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn053	*CDZ							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0001							FCV	

[Input type] Setting input
[Data type] Bit path

#1 FCV Program format
0: Series 16 standard format
1: Series 15 format

NOTE
1 Programs created in the Series 15 program format can be used for operation on the following functions:
  1 Subprogram call M98
  2 Thread cutting G32 (T series)
  3 Canned cycle G90, G92, G94 (T series)
  4 Multiple repetitive canned cycle
    G71 to G76 (T series) / G71.7 to G71.6 (M series)
  5 Canned cycle for drilling
    G83.1, G80 to G89 (T series) / G73, G74, G76, G80 to G89 (M series)
2 When the program format used in the Series 15 is used for this CNC, some limits may be added to the range of command values etc. Refer to the Operator's Manual.

	#7	#6	#5	#4	#3	#2	#1	#0
1403	RTV			ROC				

[Input type] Parameter input
[Data type] Bit path

#4 ROC In the threading cycles G92 (T series), G76 (T series), and G76.7 (M series), rapid traverse override for retraction after threading is finished is:
0: Effective
1: Not effective (Override of 100%)

#7 RTV Rapid traverse override while the tool is retracting in threading
0: Effective.
1: Not effective.

1466	Feedrate for retraction in threading cycle G92 or G76
	Feedrate for retraction in threading cycle G76.7

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
When threading cycle G92, G76 or G76.7 is specified, retraction is performed after threading. Set a feedrate for this retraction.

> ⚠ **WARNING**
> When the manual handle interruption is valid, set the same value as the parameter No.1430 to the parameter No.1466.

> **NOTE**
> When this parameter is set to 0 or bit 1 (CFR) of parameter No. 1611 is set to 1, the rapid traverse rate set in parameter No. 1420 is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1610			THLx				CTBx	CTLx

[Input type] Parameter input
[Data type] Bit axis

#0 CTLx Acceleration/deceleration in cutting feed or dry run during cutting feed
0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied.

#1 CTBx Acceleration/deceleration in cutting feed or dry run during cutting feed
0: Exponential acceleration/deceleration or linear acceleration/ deceleration is applied. (Depending on the setting in bit 0 (CTLx) of parameter No. 1610)
1: Bell-shaped acceleration/deceleration is applied.

#5 THLx Acceleration/deceleration in threading cycles
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied.
(Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)
As the time constant and FL rate, however, the settings of parameters Nos. 1626 and 1627 for threading cycles are used.

	#7	#6	#5	#4	#3	#2	#1	#0
1611								CFR

[Input type] Parameter input
[Data type] Bit path

#0 CFR For retraction after threading in the threading cycles G92 (T series), G76 (T series), and G76.7 (M series):
0: The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No. 1626) and FL rate (parameter No. 1627).
1: The type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant.

NOTE

If this parameter is set to 1, a check is made before a retraction to see that the specified feedrate has become 0 (the delay in acceleration/deceleration has become 0). For retraction, the rapid traverse rate (parameter No. 1420) is used, regardless of the setting of parameter No. 1466. When this parameter is set to 0, parameter No. 1466 is used as the feedrate for retraction. As acceleration/deceleration used for retraction, only acceleration/deceleration after interpolation is used. Rapid traverse acceleration/deceleration before interpolation and optimum torque acceleration/deceleration are disabled.

1626	Acceleration/deceleration time constant in threading cycles for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set a time constant for acceleration/deceleration after interpolation in the threading cycles G92 (T series), G76 (T series), and G76.7 (M series) for each axis.

1627	FL rate for acceleration/deceleration in threading cycles for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set an FL rate for acceleration/deceleration after interpolation in the threading cycles G92 (T series), G76 (T series), and G76.7 (M series) for each axis. Set 0 at all times except in a special case. The FL rate is valid only for exponential acceleration/deceleration after interpolation.

	#7	#6	#5	#4	#3	#2	#1	#0
5102						QSR		

[Input type] Parameter input
[Data type] Bit path

#2 QSR Before a multiple repetitive canned cycle (G70 (T series), G70.7 (M series) to G73 (T series), G73.7 (M series)) is started, a check to see if the program contains a block that has the sequence number specified in address Q is:
0: Not made.
1: Made.
When 1 is set in this parameter and the sequence number specified in address Q is not found, the alarm PS0063 is issued and the canned cycle is not executed.

	#7	#6	#5	#4	#3	#2	#1	#0
5104						FCK		

[Input type] Parameter input
[Data type] Bit path

#2 FCK In a multiple repetitive canned cycle (G71 (T series), G71.7 (M series), G72 (T series), G72.7 (M series)), the machining profile is:

0: Not checked.
1: Checked.

The target figure specified by G71, G71.7, G72, or G72.7 is checked for the following before machining operation:

- If the start point of the canned cycle is less than the maximum value of the machining profile even when the plus sign is specified for a finishing allowance, the alarm PS0322 is issued.
- If the start point of the canned cycle is greater than the minimum value of the machining profile even when the minus sign is specified for a finishing allowance, the alarm PS0322 is issued.
- If an unmonotonous command of type I is specified for the axis in the cutting direction, the alarm PS0064 or PS0329 is issued.
- If an unmonotonous command is specified for the axis in the roughing direction, the alarm PS0064 or PS0329 is issued.
- If the program does not include a block that has a sequence number specified by address Q, the alarm PS0063 is issued. This check is made, regardless of bit 2 (QSR) of parameter No. 5102.
- If a command (G41/G42) on the blank side in tool nose radius compensation is inadequate, the alarm PS0328 is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
5105						RF2	RF1	

[Input type] Parameter input
[Data type] Bit path

#1 RF1 In a multiple repetitive canned cycle (G71 (T series), G71.7 (M series), G72 (T series), G72.7 (M series)) of type I, roughing is:

0: Performed.
1: Not performed.

> **NOTE**
> When a roughing allowance (Δi/Δk) is specified using the Series 15 program format, roughing is performed, regardless of the setting of this parameter.

#2 RF2 In a multiple repetitive canned cycle (G71 (T series), G71.7 (M series), G72 (T series), G72.7 (M series)) of type II, roughing is:

0: Performed.
1: Not performed.

> **NOTE**
> When a roughing allowance (Δi/Δk) is specified using the Series 15 program format, roughing is performed, regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5106					NT2	NT1		GFX
					NT2	NT1		

[Input type] Parameter input

[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 GFX When the function of multiple respective canned cycle and grinding canned cycle are both active, G71/G72/G73/G74 commands are:
0: Multiple respective canned cycle.
1: Grinding canned cycle.

#2 NT1 In the multiple repetitive cycle G71/G72/G73 (G-code system A), when the tool nose radius compensation G40/G41/G42 is commanded in the target figure program:
0: The alarm PS0325 is occurred.
1: No alarm is occurred. However, the tool nose radius compensation command in the target figure program is ignored.

#3 NT2 In the multiple repetitive cycle G70 (G-code system A), when the tool nose radius compensation G40/G41/G42 is commanded in the target figure program:
0: The alarm PS0325 is occurred.
1: No alarm is occurred. The tool nose radius compensation command is valid.

> **NOTE**
> Make the program by following ways to enable the tool nose radius compensation in the finishing cycle G70 by commanded the tool nose radius compensation in the target figure program.
> - The tool nose radius compensation cancel G40 is selected as the modal when the finishing cycle G70 is commanded.
> - Command G41/G42 at the first block of the target figure program (commanded by P address).
> - Command G40 at the last block of the target figure program (commanded by Q address).

	#7	#6	#5	#4	#3	#2	#1	#0
5107						OCM	ASC	ASU

[Input type] Parameter input
[Data type] Bit path

#0 ASU For G71 (T series), G71.7 (M series), G72 (T series), or G72.7 (M series), movement to the last turning start position is performed by:
0: Cutting feed.
1: Rapid traverse.

For two-cycle operation to move toward the current turning start position, this parameter selects the feed in the first cycle (movement to the last turning start position). The feed in the second cycle (movement from the last turning start position to the current turning start position) follows the feed in the first block of the shape program.
This parameter is valid to both of type-I and type-II commands.

#1 ASC The G71 (T series), G71.7 (M series), G72 (T series), or G72.7 (M series) TYPE1 commands execute the movement toward the current turning start position in:
0: Two cycles.

1: One cycle.

You can change the two-cycle operation to move to the current turning start position from two cycles to one cycle. The feed mode follows the mode (G00, G01) in the first block of the shape program. This parameter is valid only to type-I commands.

#2 OCM In G70-G73 (G-code system A), the cycle operation is executed by :
0: Conventional method.
1: Improved method.

	#7	#6	#5	#4	#3	#2	#1	#0
5108			NIC		NSP		DTP	R16
			NIC		NSP		DTP	R16

[Input type] Parameter input
[Data type] Bit path

#0 R16 In the cutting up movement of the multiple repetitive cycle G71/G72 (G-code system A) of type II, if there is the block that commands just the movement of the first axis on the plane in the finishing shape program:
0: The cutting up movement is executed before the cutting of the first axis on the plane.
1: The cutting up movement is not executed and the cutting is continued along the finishing shape of the first axis on the plane.

#1 DTP In the multiple repetitive cycle G71/G72 (G-code system A) of type I, after rough cutting of the finishing shape program is finished, the tool return to the cycle start point:
0: After the tool moves to (cycle start point + distance of the finishing allowance) in order X-axis, Z-axis.
1: Directly from the end point of the finishing program.

#3 NSP In the multiple repetitive cycle G71/G72 (G-code system A) of type II, the cutting is executed:
0: By conventional path. (The same cutting path might be executed.)
1: Not to repeat the same cutting path.

> **NOTE**
> This parameter is Automatic setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

#5 NIC In Multiple repetitive cycle G71/G72 (G-code system A), in-position check between cutting blocks is:
0: Executed.
1: Not executed.

	#7	#6	#5	#4	#3	#2	#1	#0
5109					TFH	TAE	CCI	DSA
					TFH	TAE		DSA

[Input type] Parameter input
[Data type] Bit path

#0 DSA When an axis, which is not included in the specified plane, is commanded in the multiple repetitive cycle(G70-G76 (T series),G70.7-G76.7 (M series)) :
0: An alarm does not occur.
1: An alarm PS0021,"ILLEGAL PLANE SELECT" occurs.

#1 CCI In the canned cycle for turning(G90,G92,G94) (T series), the address of the taper amount :
0: Is dependent on the setting of bit 1 (FCV) of parameter No.0001.
1: Can be specified with I,J,K and R.

The address for the taper amount is dependent on the setting of the parameter of the table below.

Bit 1 (FCV) of parameter No.0001	Bit 1 (CCI) of parameter No.5109	Address for the taper amount
0	0	R
1	0	I,J,K
0	1	I,J,K and R
1	1	I,J,K and R

#2 TAE When the Series 15 format is used (with bit 1 (FCV) of parameter No.0001 set to 1), address E of threading is :
0: Inch threading (The number of threads per inch).
1: Thread lead.

NOTE
1 This parameter setting is valid when bit 1 (FCV) of parameter No.0001 is set to 1.
2 This parameter setting is available for the following G codes.

G code in the Machining system	G code In the lathe system			Function
	A	B	C	
G33	G32	G33	G33	Threading
G34	G34	G34	G34	Variable lead threading
G35	G35	G35	G35	Circular threading CW
G36	G36	G36	G36	Circular threading CCW
G76.7	G76	G76	G78	Multiple-thread cutting cycle
-	G92	G78	G21	Threading cycle

#3 TFH Behavior of feed hold in the threading cycles and multiple threading cycle is:
0: Conventional specification.
1: FS16*i* compatible specification.

	#7	#6	#5	#4	#3	#2	#1	#0
5124								CBR

[Input type] Parameter input
[Data type] Bit path

#0 CBR In a TYPE-1 of multiple repetitive canned cycle (G71 (T series), G71.7 (M series), G72 (T series), G72.7 (M series)), escape after rough cutting is performed by
0: Cutting feed.
1: Rapid traverse.(The tool path follows the setting in bit 1 (LRP) of parameter No. 1401.)

	#7	#6	#5	#4	#3	#2	#1	#0
5125						PRS		
						PRS		

[Input type] Parameter input
[Data type] Bit path

#2 PRS In Pattern repeating cycle G73 (G-code system A), the stop positions of single block operation are:
0: The end point of each cycles and the end point of each blocks in the finishing shape.
1: The end point of each cycles and the end point of escape from the cycle start point. (FS16i compatible specification)

5130	Cutting value (chamfering value) in thread cutting cycles G92 and G76
	Cutting value (chamfering value) in thread cutting cycle G76.7

[Input type] Parameter input
[Data type] Byte path
[Unit of data] 0.1
[Valid data range] 0 to 127

This parameter sets a cutting value (chamfering value) in the thread cutting cycle (G76/G76.7) of a multiple repetitive canned cycle and in the thread cutting cycle (G92) of a canned cycle.
Let L be a lead. Then, a cutting value range from 0.1L to 12.7L is allowed.
To specify a cutting value of 10.0L, for example, specify 100 in this parameter.

5131	Cutting angle in thread cutting cycles G92 and G76
	Cutting angle in thread cutting cycle G76.7

[Input type] Parameter input
[Data type] Byte path
[Unit of data] deg
[Valid data range] 1 to 89

This parameter sets a thread cutting angle in a thread cutting cycle (G92/G76/G76.7).
When 0 is set, an angle of 45 degrees is specified.

5132	Depth of cut in multiple repetitive canned cycles G71 and G72
	Depth of cut in multiple repetitive canned cycles G71.7 and G72.7

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the depth of cut in multiple repetitive canned cycles G71 and G72 or G71.7 and G72.7.
This parameter is not used with the Series 15 program format.

NOTE
Specify a radius value at all times.

5133	Escape in multiple repetitive canned cycles G71 and G72
	Escape in multiple repetitive canned cycles G71.7 and G72.7

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the escape in multiple repetitive canned cycles G71 and G72 or G71.7 and G72.7.

> **NOTE**
> Specify a radius value at all times.

5134	Clearance value in multiple repetitive canned cycles G71 and G72
	Clearance value in multiple repetitive canned cycles G71.7 and G72.7

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets a clearance value up to the cutting feed start point in multiple repetitive canned cycles (G71/G72 or G71.7/G72.7).

> **NOTE**
> Specify a radius value at all times.

5135	Retraction distance in the multiple repetitive canned cycle G73 (second axis on the plane)
	Retraction distance in the multiple repetitive canned cycle G73.7 (second axis on the plane)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets a retraction distance along the second axis on the plane in the multiple repetitive canned cycle G73 or G73.7. This parameter is not used with the Series 15 program format.

> **NOTE**
> Specify a radius value at all times.

5136	Retraction distance in the multiple repetitive canned cycle G73 (first axis on the plane)
	Retraction distance in the multiple repetitive canned cycle G73.7 (first axis on the plane)

[Input type] Parameter input
[Data type] Real path

[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets a retraction distance along the first axis on the plane in the multiple repetitive canned cycle G73 or G73.7. This parameter is not used with the Series 15 program format.

> **NOTE**
> Specify a radius value at all times.

5137	Number of divisions in the multiple repetitive canned cycle G73
	Number of divisions in the multiple repetitive canned cycle G73.7

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Cycle
[Valid data range] 1 to 99999999
This parameter sets the number of divisions in the multiple repetitive canned cycle G73 or G73.7.
This parameter is not used with the Series 15 program format.

5139	Return in multiple repetitive canned cycles G74 and G75
	Return in multiple repetitive canned cycles G74.7 and G75.7

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the return in multiple repetitive canned cycles G74 and G75 or G74.7 and G75.7.

> **NOTE**
> Specify a radius value at all times.

5140	Minimum depth of cut in the multiple repetitive canned cycle G76
	Minimum depth of cut in the multiple repetitive canned cycle G76.7

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets a minimum depth of cut in the multiple repetitive canned cycle G76 or G76.7 so that the depth of cut does not become too small when the depth of cut is constant.

> **NOTE**
> Specify a radius value at all times.

5141	Finishing allowance in the multiple repetitive canned cycle G76
	Finishing allowance in the multiple repetitive canned cycle G76.7

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the finishing allowance in multiple repetitive canned cycle G76 or G76.7.

> **NOTE**
> Specify a radius value at all times.

5142	Repetition count of final finishing in multiple repetitive canned cycle G76
	Repetition count of final finishing in multiple repetitive canned cycle G76.7

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Cycle
[Valid data range] 1 to 99999999
This parameter sets the number of final finishing cycle repeats in the multiple repetitive canned cycle G76 or G76.7.
When 0 is set, only one final finishing cycle is executed.

5143	Tool nose angle in multiple repetitive canned cycle G76
	Tool nose angle in multiple repetitive canned cycle G76.7

[Input type] Parameter input
[Data type] Byte path
[Unit of data] deg
[Valid data range] 0, 29, 30, 55, 60, 80
This parameter sets the tool nose angle in multiple repetitive canned cycle G76 or G76.7.
This parameter is not used with the Series 15 program format.

5145	Allowable value 1 in multiple repetitive canned cycles G71 and G72
	Allowable value 1 in multiple repetitive canned cycles G71.7 and G72.7

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

If a monotonous command of type I or II is not specified for the axis in the roughing direction, the alarm PS0064 or PS0329 is issued. When a program is created automatically, a very small unmonotonous figure may be produced. Set an unsigned allowable value for such an unmonotonous figure. By doing so, G71 and G72 or G71.7 and G72.7 cycles can be executed even in a program including an unmonotonous figure.

[Example] Suppose that a G71 or G71.7 command where the direction of the cutting axis (X-axis) is minus and the direction of the roughing axis (Z-axis) is minus is specified. In such a case, when an unmonotonous command for moving 0.001 mm in the plus direction along the Z-axis is specified in a target figure program, roughing can be performed according to the programmed figure without an alarm by setting 0.001 mm in this parameter.

NOTE

A check for a monotonous figure is made at all times during G71 and G72 or G71.7 and G72.7 cycles. A figure (programmed path) is checked. When tool nose radius compensation is performed, a path after compensation is checked. When bit 2 (FCK) of parameter No. 5104 is set to 1, a check is made before G71, G72, G71.7, or G72.7 cycle operation. In this case, not a path after tool nose radius compensation but a programmed path is checked. Note that no alarm is issued when an allowable value is set.
Use a radius value to set this parameter at all times.

5146	**Allowable value 2 in multiple repetitive canned cycles G71 and G72**
	Allowable value 2 in multiple repetitive canned cycles G71.7 and G72.7

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to cut of depth

If a monotonous command of type I is not specified for the axis in the cutting direction, the alarm PS0064 or PS0329 is issued. When a program is created automatically, a very small unmonotonous figure may be produced. Set an unsigned allowable value for such an unmonotonous figure. By doing so, G71 and G72 or G71.7 and G72.7 cycles can be executed even in a program including an unmonotonous figure.
The allowable value is clamped to the depth of cut specified by a multiple repetitive canned cycle.

[Example] Suppose that a G71 or G71.7 command where the direction of the cutting axis (X-axis) is minus and the direction of the roughing axis (Z-axis) is minus is specified. In such a case, when an unmonotonous command for moving 0.001 mm in the minus direction along the X-axis is specified in a target figure program for moving from the bottom of cutting to the end point, roughing can be performed according to the programmed figure without an alarm by setting 0.001 mm in this parameter.

NOTE

A check for a monotonous figure is made at all times during G71 and G72 or G71.7 and G72.7 cycles. A figure (programmed path) is checked. When tool nose radius compensation is performed, a path after compensation is checked. When bit 2 (FCK) of parameter No. 5104 is set to 1, a check is made before G71, G72, G71.7, or G72.7 cycle operation. In this case, not a path after tool nose radius compensation but a programmed path is checked. Note that no alarm is issued when an allowable value is set. Use a radius value to set this parameter at all times.

Alarm and message

Number	Message	Description
PS0061	P OR Q COMMAND IS NOT IN THE MULTIPLE REPETIVE CYCLES BLOCK	Address P or Q is not specified in multiple repetitive cycle (G70, G71, G72, or G73) command.
PS0062	THE CUTTING AMOUNT IS ILLEGAL IN THE ROUGH CUTTING CYCLE	A zero or a negative value was specified in a multiple repetitive canned rough-cutting cycle (G71 or G72) as the depth of cut.
PS0063	THE BLOCK OF A SPECIFIED SEQUENCE NUMBER IS NOT FOUND	The sequence number specified by addresses P and Q in multiple repetitive cycle (G70, G71, G72, or G73) command cannot be searched.
PS0064	THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(FIRST AXES)	In a shape program for the multiple repetitive canned rough-cutting cycle (G71 or G72), the command for the first plane axis was not a monotonous increase or decrease.
PS0065	G00/G01 IS NOT IN THE FIRST BLOCK OF SHAPE PROGRAM	In the first block of the shape program specified by P of the multiple repetitive canned cycle (G70, G71, G72, or G73), G00 or G01 was not specified.
PS0066	UNAVAILABLE COMMAND IS IN THE MULTIPLE REPETIVE CYCLES BLOCK	An unavailable command was found in a multiple repetitive canned cycle (G70, G71, G72, or G73) command block.
PS0067	THE MULTIPLE REPETIVE CYCLES IS NOT IN THE PART PROGRAM STORAGE	A multiple repetitive canned cycle (G70, G71, G72, or G73) command is not registered in a tape memory area.
PS0069	LAST BLOCK OF SHAPE PROGRAM IS AN ILLEGAL COMMAND	In a shape program in the multiple repetitive canned cycle (G70, G71, G72, or G73), a command for the chamfering or corner R in the last block is terminated in the middle.
PS0315	ILLEGAL NOSE ANGLE COMMAND IS IN THE THREAD CUTTING CYCLE	An invalid tool tip angle is specified in a multiple repetitive canned threading cycle (G76).
PS0316	ILLEGAL CUTTING AMOUNT IS IN THE THREAD CUTTING CYCLE	A minimum depth of cut higher than the thread height is specified in a multiple repetitive canned threading cycle (G76).
PS0317	ILLEGAL THREAD COMMAND IS IN THE THREAD CUTTING CYCLE	A zero or a negative value is specified in a multiple repetitive canned threading cycle (G76) as the thread height or the depth of cut.
PS0318	ILLEGAL RELIEF AMOUNT IS IN THE DRILLING CYCLE	Although an escape directions is set in a multiple repetitive canned cutting-off cycle (G74 or G75), a negative value is specified for Δd.
PS0319	THE END POINT COMMAND IS ILLEGAL IN THE DRILLING CYCLE	Although the Δi or Δk travel distance is set to 0 in a multiple repetitive canned cutting-off cycle (G74 or G75), a value other than 0 is specified for a U or W.
PS0320	ILLEGAL MOVEMENT AMOUNT/CUTTING AMOUNT IS IN THE DRILLING CYCLE	A negative value is specified in a multiple repetitive canned cutting-off cycle (G74 or G75) as Δi or Δk (travel distance/the depth of cut).

Number	Message	Description
PS0321	ILLEGAL REPEATED TIME IS IN THE PATTERN REPEATING CYCLE	A zero or a negative value is specified in a multiple repetitive canned closed loop cycle (G73) as a repeated time.
PS0322	FINISHING SHAPE WHICH OVER OF STARTING POINT	An invalid shape which is over the cycle starting point is specified in a shape program for a multiple repetitive canned rough-cutting cycle (G71 or G72).
PS0323	THE FIRST BLOCK OF SHAPE PROGRAM IS A COMMAND OF TYPE II	Type II is specified in the first block of the shape program specified by P in a multiple repetitive canned rough-cutting cycle (G71 or G72). Z (W) command is for G71. X (U) command is for G72.
PS0324	THE INTERRUPTION TYPE MACRO WAS DONE IN THE MULTIPLE REPETIVE CYCLES	An interruption type macro was issued during the multiple repetitive canned cycle (G70, G71, G72, or G73).
PS0325	UNAVAILABLE COMMAND IS IN SHAPE PROGRAM	1) Unusable command was issued in a shape program for a multiple repetitive cycle (G70, G71, G72, or G73). 2) In the multiple repetitive cycle (G70), when the tool nose radius compensation can be commanded in the target figure program (parameter NT2(No.5106#3)=1), G41 or G42 is commanded at the target figure program other than the first block.
PS0326	LAST BLOCK OF SHAPE PROGRAM IS A DIRECT DRAWING DIMENSIONS	In a shape program in the multiple repetitive canned cycle (G70, G71, G72, or G73), a command for direct input of drawing dimensions in the last block is terminated in the middle.
PS0327	MODAL THAT MULTIPLE REPETIVE CYCLES CANNOT BE DONE	A multiple repetitive canned cycle (G70, G71, G72, or G73) was commanded in a modal state in which a multiple repetitive canned cycle could not be commanded.
PS0328	ILLEGAL WORK POSITION IS IN THE TOOL NOSE RADIUS COMPENSATION	The specification for the blank side for a tool nose radius compensation (G41 or G42) is incorrect in a multiple repetitive canned cycle (G71 or G72).
PS0329	THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(SECOND AXES)	In a shape program for the multiple repetitive canned rough-cutting cycle (G71 or G72), the command of the second plane axis was not a monotonous increase or decrease.
PS0330	ILLEGAL AXIS COMMAND IS IN THE TURNING CANNED CYCLE	An axis other than the plane is specified n a canned cycle(G90, G92, or G94).

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Cutting feed
OPERATOR'S MANUAL For Lathe System (B-64694EN-1)	Threading (G32)
	Multiple threading
	Canned cycle
	Multiple repetitive canned cycle
OPERATOR'S MANUAL For Machining Center System (B-64694EN-2)	Threading (G33)
	Multiple threading
	Multiple repetitive canned cycle

12.8.2 Reducing path for Multiple repetitive canned cycle

T

Overview

In multiple repetitive canned cycle, a path of cycle can be reduced.

Explanation

In stock removal in turning (G71, G71.7) of multiple repetitive canned cycle, conventional path was as follows (Fig. 12.8.2 (a)Fig. 12.8.2 (a)Fig. 12.8.2 (a)).

1. Moving in the direction of the first axis on the plane (Z-axis for the ZX plane)
2. Moving in the direction of the second axis on the plane (X-axis for the ZX plane)

In this function, it directly moves from the retraction end point to the last turning start point (Fig. 12.8.2 (b), Fig. 12.8.2 (c)).
The cycle time can be reduced because moving path is reduced.
To use this function, set bit 6 (RPA) of parameter No. 5125 to 1.

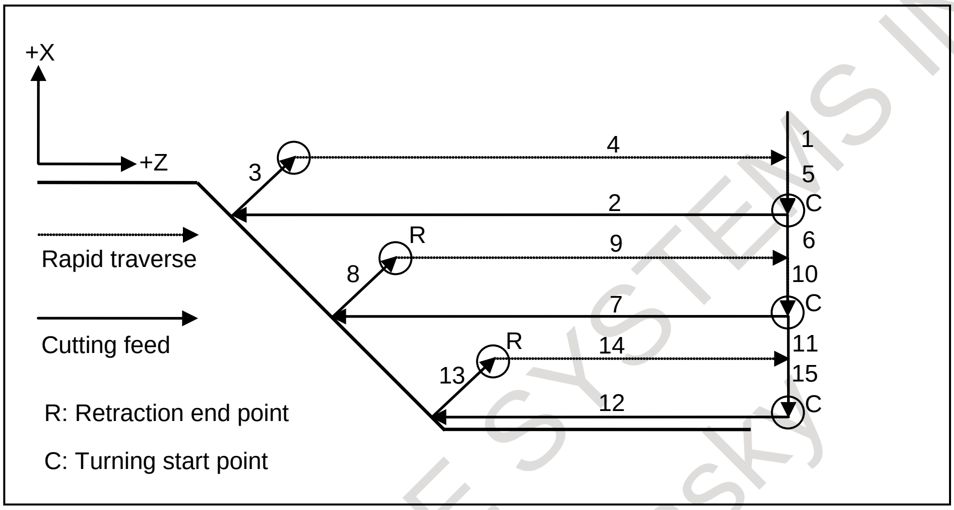


Fig. 12.8.2 (a) Conventional operation

Movement from retraction end point to last turning start point

In a start block in the program for a target figure, the movement from the retraction end point to the last turning start point is changed.

When a start block in the program for a target figure is G00 :
Axes move from the retraction end point to the last turning start point by rapid traverse.

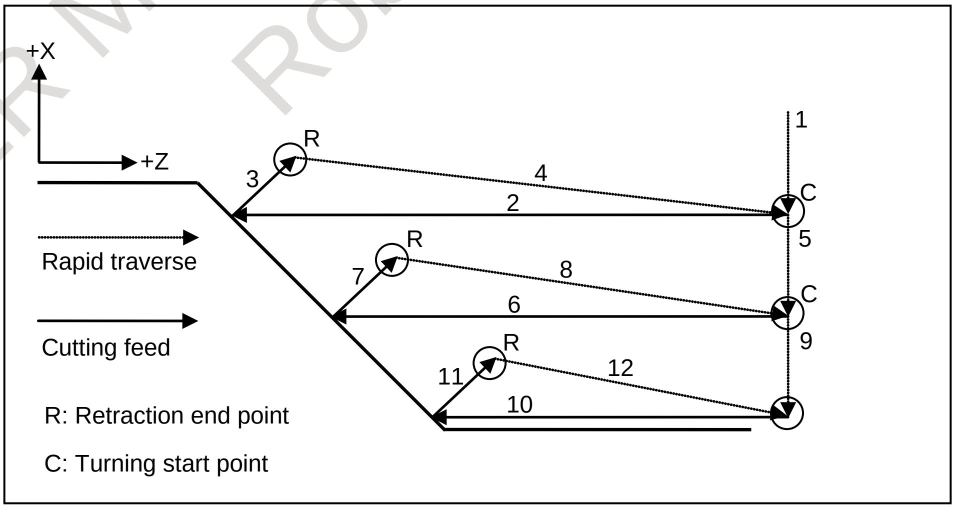


Fig. 12.8.2 (b) Operation when this function is effective (Start block in the program for a target figure is G00.)

When a start block in the program for a target figure is G01 :
Axes move from the retraction end point by rapid traverse until becoming a value that the distance to the last turning start point is set to parameter No.5126. These move from there to the last turning start point

by cutting feed. When 0 is set in parameter No.5126, this parameter is the distance that is 50% of retraction distance.

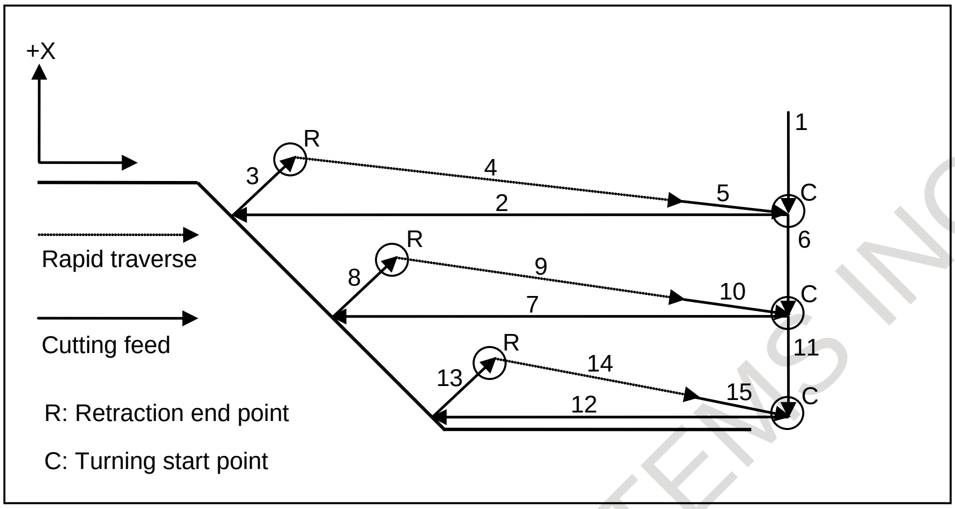


Fig. 12.8.2 (c) Operation when this function is effective (Start block in the program for a target figure is G01.)

> ⚠ **WARNING**
> Specify G01 in a start block in the program for a target figure when turning start point is near the processing surface. The distance of cutting can be adjusted by parameter No.5126.

Effect of reducing cycle time

In this function, it is more effective in reducing cycle time to assume a start block in the program for a target figure to be G00.
Therefore, it is recommended that turning start point is separated from work piece, and it specifies G00 in the start block in the program for a target figure.

Limitation

- This function is valid only when bit 1 (LRP) of parameter No. 1401 is 1.
- This function is valid only in stock removal in turning (G71, G71.7) and stock removal in facing (G72, G72.7) of multiple repetitive canned cycle.
- This function is invalid in type II of multiple repetitive canned cycle.

Parameter

Parameters which are used for this function are as follows.

	#7	#6	#5	#4	#3	#2	#1	#0
5125		RPA						

[Input type] Parameter input
[Data type] Bit path

#6 RPA In type I of G71 (T series), G71.7 (M series), G72 (T series) and G72.7 (M series) of multiple repetitive canned cycle, the axes :
0: Don't move directly from retraction end point to the last turning start point.
1: Move directly from retraction end point to the last turning start point.

> **NOTE**
> When the distance to the last turning start point reaches the value of the parameter No.5126, the movement of the axis changes from rapid traverse to cutting feed.

5126	**Distance in which cutting to the last turning start point in multiple repetitive canned cycles G71 and G72**
	Distance in which cutting to the last turning start point in multiple repetitive canned cycles G71.7 and G72.7

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to retraction distance

In type I of G71 (T series), G71.7 (M series), G72 (T series) and G72.7 (M series) of multiple repetitive canned cycle, the distance in which cutting to the last turning start point is set to this parameter. When 0 is set in parameter No.5126, this parameter is the distance that is 50% of retraction distance.
When the setting value is out of range, this parameter is the distance in which cutting to the last turning start point is the same distance as retraction distance.

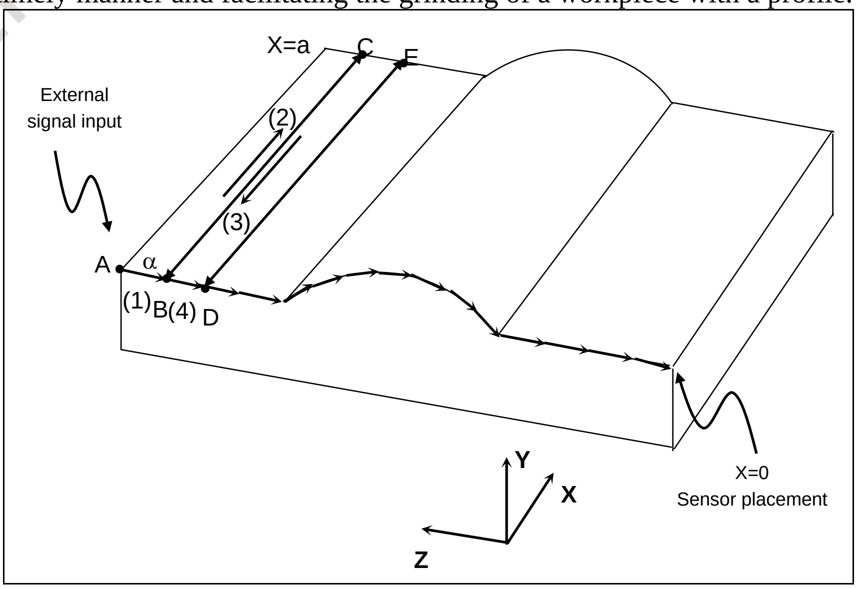
> **NOTE**
> This parameter is valid when bit 6 (RPA) of parameter No. 5125 is set to 1 and start block in the program for a target figure is G01.

12.9 IN-FEED CONTROL (FOR GRINDING MACHINE)

M

Overview

Each time an external signal is input at the table swing end point, a workpiece is cut by a constant depth of cut along a programmed figure on the specified Y-Z plane. This makes it possible to perform grinding and cutting in a timely manner and facilitating the grinding of a workpiece with a profile.



For example, it is possible to machine a workpiece with a profile programmed with linear interpolation, circular interpolation, and linear interpolation on the YZ plane, such as that shown in the figure above.
A sensor is placed at a X = 0 position so that the external signal is input when the sensor detects the grinding wheel. When the program is started at point A, the machine is first placed in the state in which it waits for the input of the external signal. Then, when the sensor detects the grinding wheel, the external signal is input, and the machine makes a cut by the constant amount α along the programmed profile on the specified YZ plane and moves to point B (operation (1)). The machine is then placed in the state in which it waits for the input of the external signal again, and performs a grinding operation along the X-axis. It grinds from point B to point C (operation (2)) and grinds back from point C to point B (operation (3)). When the machine returns to point B, the sensor detects the grinding wheel again, and the external signal is input, so that the machine makes a cut by the amount of α and moves to point D (operation (4)). At point D, the machine performs a grinding operation along the X-axis.
Afterwards, each time the external signal is input, the machine makes a cut by the amount of α along the profile program, so that the workpiece is machined to a profile such as that shown in the figure above.

Signal

In-feed control cut start signal INFD<Gn063.6>

[Classification] Input signal
[Function] Exercises in-feed control.
[Operation] When this signal is set from 0 to 1, the control unit operates as follows:
- A movement is made by a specified depth of cut along a program figure.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn063		INFD						

Alarm and message

Number	Message	Description
PS0230	R CODE NOT FOUND	Cut depth R is not specified in the block including G161. Alternatively, the value specified for R is negative. Modify the program.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (For Machining Center System) (B-64694EN-2)	In-feed control (for grinding machine)

12.10 CANNED GRINDING CYCLE (FOR GRINDING MACHINE)

Overview

With the canned grinding cycle, repetitive machining operations that are specific to grinding and are usually specified using several blocks can be specified using one block including a G function. So, a program can be created simply. At the same time, the size of a program can be reduced, and the memory can be used more efficiently. Four types of canned grinding cycles are available:

T

- Traverse grinding cycle
- Traverse direct constant-size grinding cycle
- Oscillation grinding cycle
- Oscillation direct constant-size grinding cycle

M

- Plunge grinding cycle
- Direct constant-dimension plunge grinding cycle
- Continuous-feed surface grinding cycle
- Intermittent-feed surface grinding cycle

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5101								FXY

[Input type] Parameter input
[Data type] Bit path

#0 FXY The drilling axis in the drilling canned cycle, or cutting axis in the grinding canned cycle is:

0: In case of the Drilling canned cycle:
  Z-axis at all times.
  In case of the Grinding canned cycle:
  - For the Lathe system
    Z-axis at all times.
  - For the Machining Center system
    G75,G77 command :Y-axis
    G78,G79 command :Z-axis

1: Axis perpendicular to a specified plane (G17/G18/G19).

> **NOTE**
> In the case of the T series, this parameter is valid only for the drilling canned cycle in the Series 15 format.

	#7	#6	#5	#4	#3	#2	#1	#0
5105								
		GIJ						

[Input type] Parameter input
[Data type] Bit path

#6 GIJ When a grinding canned cycle in the machining center system is executed, if the signs of I, J, and K are different:

0: An alarm is issued.
1: An operation compatible with the FS16i is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
5106								GFX

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 GFX When the function of multiple respective canned cycle and grinding canned cycle are both active, G71/G72/G73/G74 commands are:
0: Multiple repetitive canned cycle.
1: Grinding canned cycle.

5176	Grinding axis number in Traverse Grinding Cycle(G71)
	Grinding axis number in Plunge Grinding Cycle(G75)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
For the Lathe system:
Set the Grinding axis number of Traverse Grinding Cycle(G71).
For the Machining Center system:
Set the Grinding axis number of Plunge Grinding Cycle(G75).

NOTE
The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456 is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5177	Grinding axis number of Traverse direct constant-size Grinding cycle(G72)
	Grinding axis number of Direct Constant Dimension Plunge Grinding Cycle(G77)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
For the Lathe system:
Set the Grinding axis number of Traverse direct constant-size Grinding cycle(G72).
For the Machining Center system:
Set the Grinding axis number of Direct Constant Dimension Plunge Grinding Cycle (G77).

NOTE
The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456 is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5178	Grinding axis number of Oscillation Grinding Cycle(G73)
	Grinding axis number of Continuous feed surface grinding cycle(G78)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
For the Lathe system:
Set the Grinding axis number of Oscillation Grinding Cycle(G73).
For the Machining Center system:
Set the Grinding axis number of Continuous feed surface grinding cycle(G78).

> **NOTE**
> The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456 is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5179	Grinding axis number of Oscillation Direct Fixed Dimension Grinding Cycle(G74)
	Grinding axis number of Intermittent feed surface grinding cycle(G79)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

For the Lathe system:
Set the Grinding axis number of Oscillation Direct Fixed Dimension Grinding Cycle(G74).
For the Machining Center system:
Set the Grinding axis number of Intermittent feed surface grinding cycle(G79).

> **NOTE**
> The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456 is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5180	
	Axis number of dressing axis in Plunge grinding cycle(G75)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the axis number of dressing axis in Plunge grinding cycle(G75).

> **NOTE**
> The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456 is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

5181	
	Axis number of dressing axis in Direct constant dimension plunge grinding cycle(G77)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the axis number of dressing axis in Direct constant dimension plunge grinding cycle(G77).

> **NOTE**
> The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456 is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

5182	Axis number of dressing axis in Continuous feed surface grinding cycle(G78)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the axis number of dressing axis in Continuous feed surface grinding cycle(G78).

> **NOTE**
> The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456 is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

5183	Axis number of dressing axis in Intermittent feed surface grinding cycle(G79)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the axis number of dressing axis in Intermittent feed surface grinding cycle(G79).

> **NOTE**
> The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456 is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

Alarm and message

Number	Message	Description
PS0370	G31P/G04Q ERROR	1) The specified address P value for G31 is out of range. The address P range is 1 to 4 in a multistage skip function. 2) The specified address Q value for G04 is out of range. The address Q range is 1 to 4 in a multistage skip function. 3) P1-4 for G31, or Q1-4 for G04 was commanded without a multistage skip function. 4) <T series > The specified value of address P of G72 or G74 falls outside the range. Address P ranges from 1 to 4 in the multistage skip function. P1-4 was specified in G72 or G74 even though the multistage skip function is not present.

Number	Message	Description
PS0455	ILLEGAL COMMAND IN GRINDING	In grinding canned cycles: 1) <M series> The signs of the I, J, and K commands do not match. 2) <M series/T series > The amount of travel of the grinding axis is not specified.
PS0456	ILLEGAL PARAMETER IN GRINDING	Parameters related to grinding canned cycles are incorrectly set. Probable causes are given below. 1) <M series/T series> The axis number of the grinding axis is incorrectly set (parameters Nos. 5176 to 5179). 2) <M series> The axis number of the dressing axis is incorrectly set (parameters Nos. 5180 to 5183). 3) <M series/T series> The axis numbers of the cut axis, grinding axis, and dressing axis (only for the M series) overlap.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (For Lathe System) (B-64694EN-1)	Canned grinding cycle (for grinding machine)
OPERATOR'S MANUAL (For Machining Center System) (B-64694EN-2)	Canned grinding cycle (for grinding machine)

12.11 MIRROR IMAGE FOR DOUBLE TURRET

T

Overview

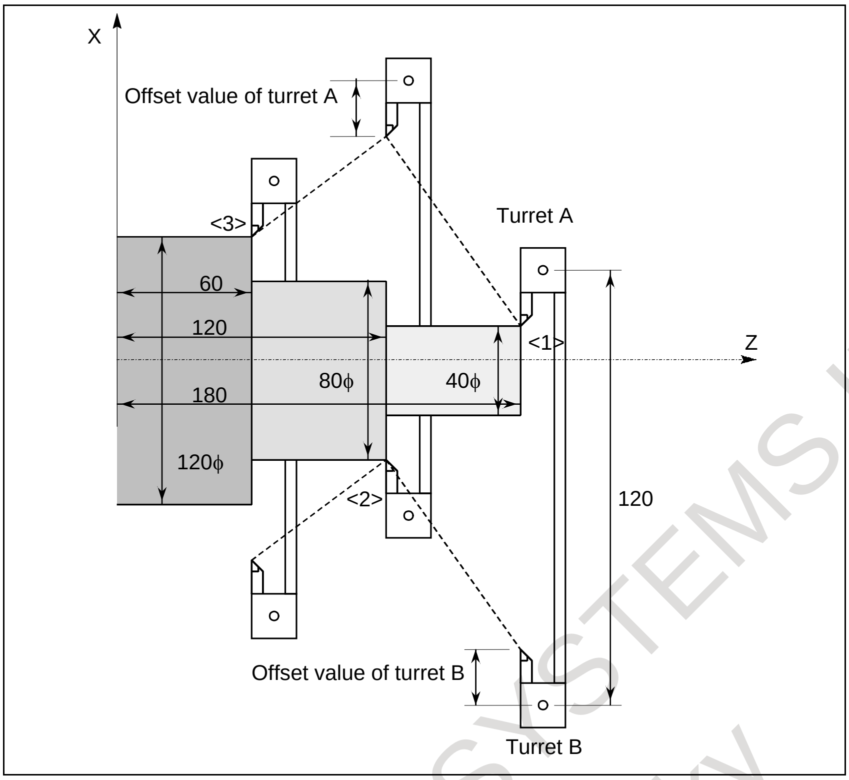
In a machine having double turrets comprising two facing turrets placed on the same control axis, a machining program for facing turrets can be created using G codes as if it existed on the same coordinate system for symmetric cutting by creating a mirror image relative to the X-axis.

G68 : Start double turret mirror image
G69 : Mirror image cancel

Explanation

A mirror image can be created using G codes relative to the X-axis of basic three axes set by parameter No. 1022. When G68 is active, the coordinate system is shifted to the other turret, and the X-axis sign is reversed from the programmed command. To use this function, set the distance between the two turrets in a parameter No. 1290.

Example) Program for double turrets



X40.0 Z180.0 T0101 ;	Position turret A at <1>
G68 ;	Shift the coordinate system by the distance A to B (120mm), and turn mirror image on.
X80.0 Z120.0 T0202 ;	Position turret B at,<2>
G69 ;	Shift the coordinate system by the distance B to A, and cancel mirror image.
X120.0 Z60.0 T0101 ;	Position turret A at <3>

* X axis is programmed diameter command.

Parameter

1290	Distance between two opposite tool posts in mirror image

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the distance between two opposite tool posts in mirror image.

Note

NOTE

1 When the G68 command based on this function is enabled, the X-coordinate value that can be read with the custom macro system variables #5041 and up or #100101 and up (current specified position (in the workpiece coordinate system)) is a position with mirror image applied.

2 This function and the balance cut function cannot be used at the same time. If both functions are active, this function is enabled or disabled depending on the system combination.
This function is enabled in:
- One path system
This function is disabled in:
- Two or more path system

3 In the following cases, an alarm is issued:
(1) When instructing of mirror image for double turret (G68) in the polar coordinate interpolation mode (alarm PS0146, "ILLEGAL USE OF G-CODE")
(2) When instructing of mirror image for double turret (G68) in the cylindrical interpolation mode (alarm PS0176, "ILLEGAL G-CODE USE(G07.1 MODE)")

Reference item

Manual name	Item name
OPERATOR'S MANUAL (For Lathe System) (B-64694EN-1)	Mirror image for double turret

12.12 INDEX TABLE INDEXING

M

Overview

By specifying indexing positions (angles) for one rotation axis (usually referred to as axis A, B, or C), the index table of the machining center can be indexed.
Before and after indexing, the index table is automatically unclamped or clamped.

NOTE
To use this function, enable Index table indexing function (bit 3 (IXC) of parameter No.8132 is 1).

Explanation

- Basic Procedure

The control axis (a rotation axis) used for index table indexing (hereafter called the index table indexing axis) is usually referred to as axis A, B, or C.
In the explanation below, the index table indexing axis is axis B. If the axis name in the actual machine is not B, the user should read axis B as referring to that axis name.
The positioning angle for the index table is commanded by the numeric following "B" in the program command, which is an independent block. Both absolute and incremental commands are possible, but the value after "B" is the integer times the numeric set by the parameter:

(Example)	G00G90B100000;	Absolute command	(Positioning angle 10 degrees)
	G00G91B20.0;	Incremental command	(Move distance 20 degrees)

There are two variations of the procedure (type A and type B) to set the index table position; the difference is in the ON/OFF timing of the position control servo. The sequence of events and the difference between the variations are described below.

(1) Assume Bbbbb is ordered by the command program.
(2) The CNC turns the B axis unclamp signal BUCLP <Fn061.0> to “1”. (Type B -- When BUCLP is turned to " 1 ", the position control servo for the B axis is turned ON.)
(3) On the PMC side, the clamp of the B axis is released with the signal BUCLP <Fn061.0> turns to "1"; when completed, the B axis unclamp completion signal *BEUCP <Gn038.6> turns to “0”.
(4) The CNC then turns the B axis unclamp signal BUCLP to “0” with the signal *BEUCP turns to "0", to indicate it received the *BEUCP signal.
(5) When the PMC is notified that BUCLP has been turned to 0, the PMC should turn *BEUCP to “1”. In type A, B-axis unclamp signal BUCLP is turned to “0”, B-axis position control is made in servo-on state, B-axis is rotated, and the B axis is stopped at the specified position. B axis always moves at rapid traverse.
(6) When the B axis stops at the specified position, CNC turns B-axis clamp signal BCLP <Fn061.1> to “1”. In type A, signal BCLP is set to “1” and B-axis position control is made in servo-off state.
(7) When BCLP is turned to “1” on the PMC side, the B axis is clamped mechanically. When the clamp is completed, the B axis clamp completion signal *BECLP <Gn038.7> is turned to “0”.
(8) When *BECLP is turned to “0”, the CNC then turns BCLP to “0”, informing it received the *BECLP signal. (Type B -- When BCLP turns to 0, the B axis position control servo is turned off.)
(9) On the PMC side, when BCLP changes to “0”, *BECLP is turned to “1”.

This completes the sequence.
The time charts for these operations are shown in the Fig. 12.12 (a) and Fig. 12.12 (b).

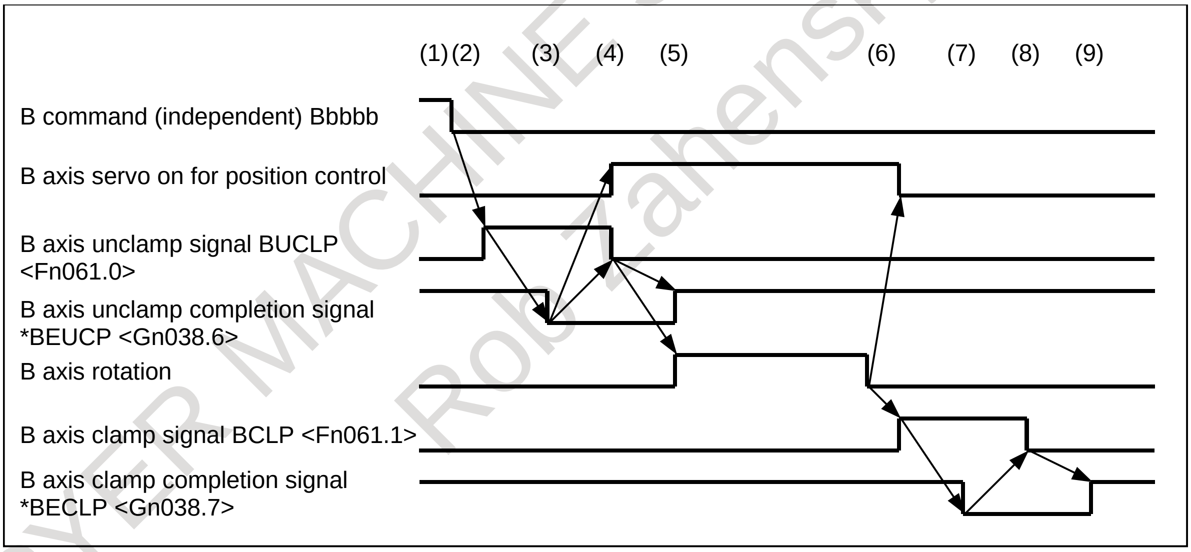


Fig. 12.12 (a) Time chart for positioning index table (type A)

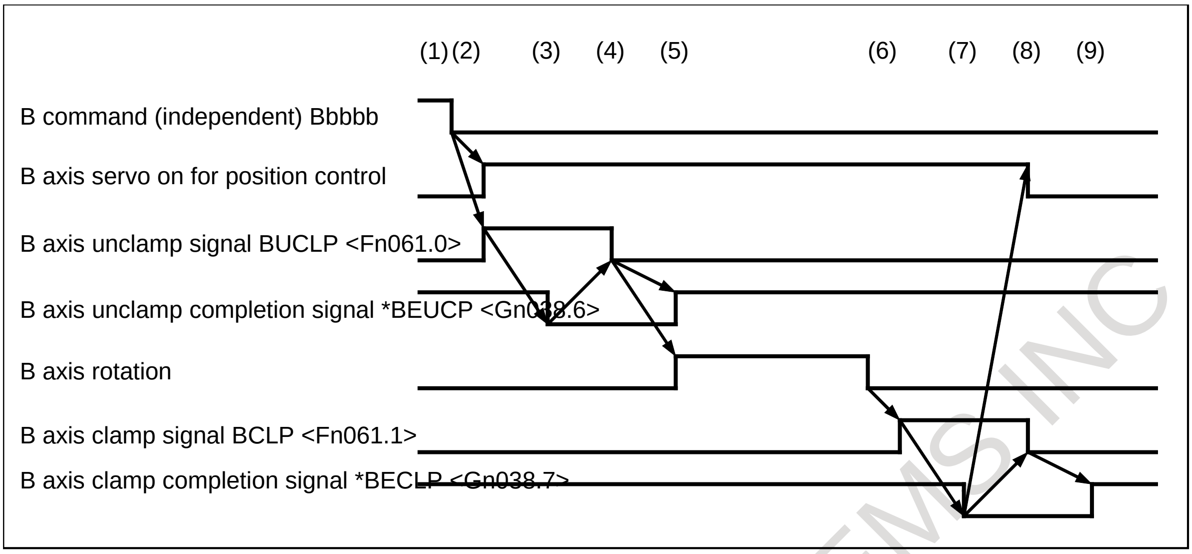


Fig. 12.12 (b) Time chart for positioning index table (type B)

The Fig. 12.12 (c) shows the timing chart for type-A manual reference position return of the B axis.

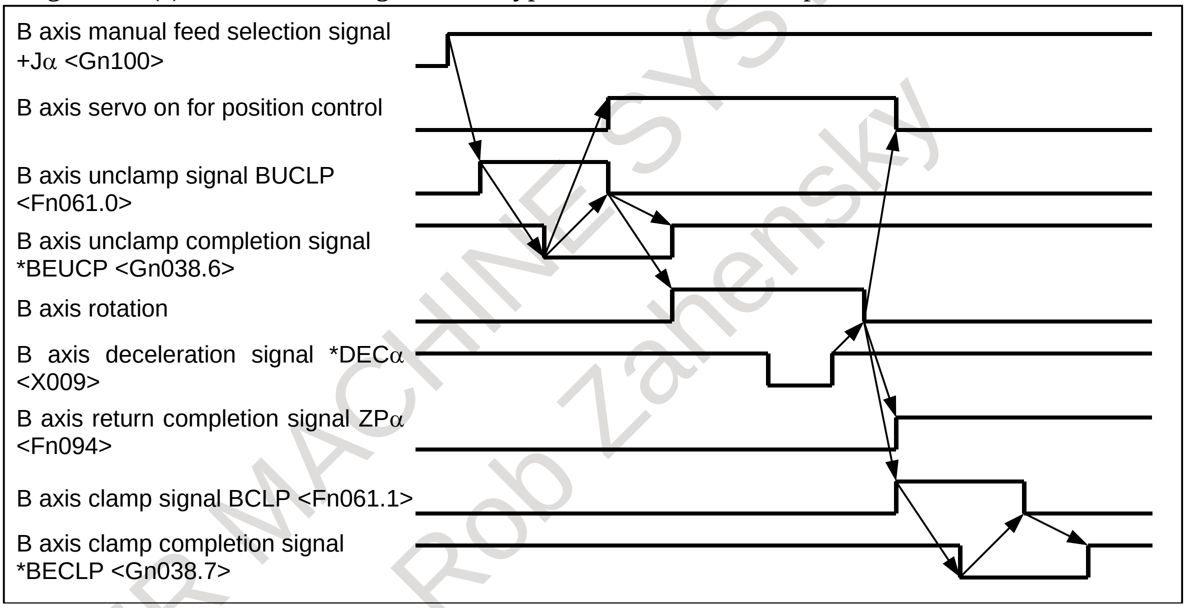


Fig. 12.12 (c) Manual reference position return of B axis time chart (type A)

- Type A and Type B

As described in the basic procedure, type A differs from type B in that the servo used for B-axis position control is turned on or off at the different timing.
Type A is suitable for a system in which the B-axis is clamped at some decided angles with the one like shot pins.
Type B is suitable for a system in which the B-axis is clamped at arbitrary angles with the one like a clutch.

- Minimum indexing angle

The minimum index table indexing angle set in parameter No. 5512 is used. An integral multiple of a set value can be specified as an indexing angle. If a value other than an integral multiple is specified, an alarm PS1561 is issued.

- Setting of an index table indexing axis

Be sure to set a rotation axis (A type) as an index table indexing axis.
(Set bit 0 (ROTx) of parameter No. 1006 to 1 and set bit 1 (ROSx) of parameter No. 1006 to 0.)

Be sure to set a non roll-over invalidity axis as an index table indexing axis.
(Set bit 0 (ROAx) of parameter No. 1008 to 0.)

- Absolute/incremental programming

Setting bit 4 (G90) of parameter No .5500, specifies absolute programming, and override the G90/G91 G-codes.

- Rotation direction

If a value other than 0 is set in the M code for specifying negative direction rotation (parameter No. 5511), movement in the negative direction is made only when a move command is specified together with the M code. In this case, movement is performed in the negative direction, regardless of whether absolute/incremental programming is used.
If 0 is set in the M code for specifying negative direction rotation (parameter No. 5511), the rotation direction in G90 mode is determined by bit 3 (INC) of parameter No. 5500, to short-cut the rotation.

- Feedrate

The table is always rotated around the indexing axis in rapid traverse mode.
Dry run cannot be executed for the indexing axis.

- Reset

If a reset is performed in the clamp or unclamp completion wait state, the clamp or unclamp signal is cleared, and the CNC exits from the completion wait state.

- Index table indexing function and other functions

Item	Explanation
Relative position display	This value can be rounded by setting bit 1 (REL) of parameter No. 5500.
Absolute position display	This value can be rounded by setting bit 2 (ABS) of parameter No. 5500.
Single direction positioning (G60)	Not specifiable
Skip function (G31)	Not specifiable
Second auxiliary function	Ensure that a second auxiliary function axis name does not duplicate an indexing axis name.
Operation during index table indexing axis movement	Unless otherwise processed by the machine, feed hold, interlock, and emergency stop can be executed during index table indexing axis movement. Machine lock can be executed after indexing has been completed.
Servo-off signal	Disable the servo-off signal for the index table indexing axis. That is, set bit 0 (FUPx) of parameter No. 1819 to 1. Usually, the index table indexing axis is in the servo-off state.
Incremental command for the index table indexing axis	When incremental programming is used for index table indexing (when bit 4 (G90) of parameter No. 5500 is set to 0), the workpiece origin offset of the index table indexing axis must always be 0. That is, there must always be a match between the workpiece coordinate system and machine coordinate system of an index table indexing axis.
Operation for an index table indexing axis	Operation in JOG/INC/HANDLE mode for an index table indexing axis is disabled. However, manual reference position return is possible. If the axis selection signal is set to 0 during manual reference position return, the movement stops immediately, and the clamp command is not executed.

NOTE

Setting for an index table indexing axis is set by parameter No. 5500 and over described as "Parameter - Setting for positioning the index table" later.

Limitation

- Simultaneous specification together with other controlled axes

If an index table indexing axis and other controlled axes are specified in the same block:

- When the command is to be executed with all axes:
  Set bit 6 (SIM) of parameter No. 5500 to 1.
- When axes for which the command is executed are to be selected:
  Set bit 6 (SIM) of parameter No. 5500 to 0. Next, set bit 0 (IXS) of parameter No. 5502 to 1 for other controlled axes to be selected in the same block.

If an index table indexing axis and other controlled axes are specified in the same block in a case other than the above, alarm PS1564 is issued.

Simultaneous specification together with other controlled axes (when G00, G28, or G30 (or G00 mode) is set)

	SIM = 0	SIM = 1
Axis with IXS = 0	Alarm PS1564	The command is executed for all axes.
Axis with IXS = 1	The command is executed.	

If an index table indexing axis and other controlled axes are specified in the same block when a command other than G00, G28, and G30 (or G00 mode) is specified, alarm PS1564 is issued.

- Command specifying zero move amount

When the amount of movement is 0, a clamp/unclamp operation is not performed.

- Look-ahead acceleration/deceleration before interpolation

In a block specifying an index table indexing axis, look-ahead acceleration/deceleration before interpolation is disabled.

- Functions that cannot be used at the same time

An axis used with any of the following functions is uncontrollable as an index table indexing axis:

- PMC axis control
- Pole position detection function
- Tilted working plane indexing
- Cs contour control
- Polar coordinate interpolation
- Cylindrical Interpolation
- Spindle control with servo motor
- High precision oscillation function
- Polygon Turning
- Electronic Gear Box

The following function cannot be used for the index table indexing axis.

- Reference position setting with mechanical stopper
- Real-time custom macro
- Retrace
- Program restart

> **NOTE**
> Functions that cannot be used at the same time might be changed or added by adding new CNC function.

Caution

⚠ CAUTION

1. While the index table is being positioned, input signals that reset the CNC, such as emergency stop signal *ESP <X008.4>, (external reset signal) ERS <Gn008.7>, and reset & rewind signal RRW <Gn008.6>, are functional.
   When reset is applied to the CNC, indexing stops. Further, if automatic operation stop signal *SP <Gn008.5> turns to 0, axis movement is stopped and the equipment enters the automatic operation stop state.
   If a stop at any position is not suitable for the machine, appropriate processing is required on the machine.
2. If the axis selection signal is set to 0 during manual reference position return, movement is stopped immediately, and the clamp command is not executed. If this proves inconvenient, measures must be taken on the machine side so that, after the axis selection signal is set to 1, it is not set to 0 until reference position return is completed.
3. When an incremental command is used for indexing the index table, the workpiece origin offset for the index table indexing axis must always be 0. This means that the workpiece coordinate system and machine coordinate system of the index table indexing axis must always match.
4. If a reset is made during indexing of the index table, a reference position return must be made before the index table is indexed subsequently.
5. The secondary auxiliary function can be used, but its address must be different from that of the indexing axis.
6. For a path on which the index table indexing function is not to be used, disable the index table indexing function (set bit 0 (ITI) of parameter No. 5501 to 0).

Note

NOTE

For positioning on an index table indexing axis, the dry run signal DRN <Gn046.7> has no effect.

Signal

B axis clamp signal BCLP <Fn061.1>

[Classification] Output signal
[Function] Instructs the PMC side to clamp the B axis mechanically.
[Output cond.] The output condition and procedure are the same as those described in the basic procedure for positioning the index table.

B axis clamp completion signal *BECLP <Gn038.7>

[Classification] Input signal
[Function] Notifies the CNC of completion of the B axis clamp operation.
[Operation] The output condition and procedure are the same as those described in the basic procedure for positioning the index table.

B axis unclamp signal BUCLP <Fn061.0>

[Classification] Output signal
[Function] Instructs the PMC side to release the B axis from the mechanical clamp.
[Output cond.] The output condition and procedure are the same as those described in the basic procedure for positioning the index table.

B axis unclamp completion signal *BEUCP <Gn038.6>

[Classification] Input signal

[Function] Notifies the CNC of completion of the release of the B axis from the mechanical clamp.

[Operation] The output condition and procedure are the same as those described in the basic procedure for positioning the index table.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn038	*BECLP	*BEUCP						

	#7	#6	#5	#4	#3	#2	#1	#0
Fn061							BCLP	BUCLP

Parameter

- Setting linear or rotation axis

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

[Input type] Parameter input

[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx Setting linear or rotation axis.
#1 ROSx

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No. 1008. (Bits 2 (ABS) and 1 (REL) of parameter No. 5500 are used for an index table indexing axis.) (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

- Setting for positioning the index table

	#7	#6	#5	#4	#3	#2	#1	#0
5500	IDX	SIM		G90	INC	ABS	REL	DDP

[Input type] Parameter input
[Data type] Bit path

#0 DDP As the method for inputting a decimal point in a command for the index table indexing axis:
0: The conventional method is used.
1: The pocket calculator method is used.

#1 REL The position display of the index table indexing axis in the relative coordinate system is:
0: Not rounded by one rotation.
1: Rounded by one rotation.

#2 ABS The position display of the index table indexing axis in the absolute coordinate system is:
0: Not rounded by one rotation.
1: Rounded by one rotation.

NOTE
Be sure to set this parameter to 1.
If an absolute programming is commanded to the index table indexing axis when this parameter is set to 0, the position display of the absolute coordinate system may be not corresponding to the absolute programming value like the following example.

Example) When indexing by rotating in a positive direction :
N10 G90 B20.;
N20 B10.; ← It rotates in a positive direction by 350 degree. At this time, 370.0 is displayed to the position display of the absolute coordinate system.

#3 INC When the M code that specifies rotation in the negative direction (parameter No. 5511) is not set, rotation in the G90 mode is:
0: Not set to the shorter way around the circumference.
1: Set to the shorter way around the circumference.

#4 G90 A command for the index table indexing axis is:
0: Assumed to be an absolute or incremental programming according to the mode.
1: Always assumed to be an absolute programming.

#6 SIM When the same block includes a command for the index table indexing axis and a command for another controlled axis:
0: The setting of bit 0 (IXS) of parameter No. 5502 is followed.
1: The commands are executed.

NOTE
Even when this parameter is set to 1, an alarm PS1564, “INDEX TABLE AXIS - OTHER AXIS SAME TIME” is issued if the block is neither G00, G28, nor G30 (nor the G00 mode).

#7 IDX Operation sequence of the index table indexing axis:
0: Type A
1: Type B

	#7	#6	#5	#4	#3	#2	#1	#0
5501							ISP	ITI

[Input type] Parameter input
[Data type] Bit path

#0 ITI The index table indexing function is:
0: Enabled.
1: Disabled.

#1 ISP Servo-off for an index table indexing axis at the completion of clamping is:
0: Processed by the CNC.
1: Not processed by the CNC. (The CNC follows the status of the servo-off signal SVF1 to SVF8<Gn0126> input from the PMC.)

	#7	#6	#5	#4	#3	#2	#1	#0
5502								IXSx

[Input type] Parameter input
[Data type] Bit axis

#0 IXSx When a command is specified in a block that contains a command for the index table indexing axis:
0 : An alarm PS1564, “INDEX TABLE AXIS - OTHER AXIS SAME TIME” is issued.
1 : The command is executed.

If bit 6 (SIM) of parameter No. 5500 is set to 1, a simultaneous operation with all axes except the index table indexing axis can be performed regardless of the setting of this parameter.
To set an axis that allows simultaneous operation for each axis, set SIM to 0, and set this parameter.

> **NOTE**
> Even when this parameter is set to 1, an alarm PS1564, “INDEX TABLE AXIS - OTHER AXIS SAME TIME” is issued if the block is neither G00, G28, nor G30 (nor the G00 mode).

- Setting of an index table indexing axis

5510	Controlled axis number of the index table indexing axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path

[Valid data range] 0 to Number of controlled axes

This parameter sets the number of a controlled axis to be used as the index table indexing axis.

When 0 is set, the fourth axis is assumed.

If the system does not have the fourth axis or the index table indexing function is not used, set bit 0 (ITI) of parameter No. 5501 to 1 to disable the function.

- Setting of a negative direction rotation command M code

5511	M code that specifies rotation in the negative direction for index table indexing

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999999

0: The rotation direction for the index table indexing axis is determined according to the setting of bit 3 (INC) of parameter No. 5500 and a command.

1 to 99999999:
The rotation for the index table indexing axis is always performed in the positive direction. It is performed in the negative direction only when a move command is specified together with the M code set in this parameter.

> **NOTE**
> Be sure to set bit 2 (ABS) of parameter No. 5500 to 1.

- Setting of a minimum positioning angle for index table indexing

5512	Minimum positioning angle for the index table indexing axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the minimum positioning angle (travel distance) for the index table indexing axis. The travel distance specified in the positioning command must always be an integer multiple of this setting. When 0 is set, the travel distance is not checked.

The minimum positioning angle is checked not only for the command, but also for the coordinate system setting and workpiece origin offset.

Alarm and message

Number	Message	Contents
PS1508	DUPLICATE M-CODE (INDEX TABLE REVERSING)	The M code value (parameter No. 5511) for specifying negative direction rotation is already used by other functions. (Index table indexing)
PS1561	ILLEGAL INDEXING ANGLE	The specified angle of rotation is not an integer multiple of the minimum indexing angle.
PS1564	INDEX TABLE AXIS – OTHER AXIS SAME TIME	The index table indexing axis and another axis have been specified in the same block.
PS1567	INDEX TABLE AXIS DUPLICATE AXIS COMMAND	Index table indexing was specified during axis movement or on an axis for which the index table indexing sequence was not completed.

Note

NOTE

If an index table indexing axis and other controlled axes are specified in the same block, G00 performs nonlinear positioning. So, if rapid traverse based on look-ahead acceleration/deceleration before interpolation is set, the setting is switched automatically to acceleration/deceleration after interpolation.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (For Machining center system) (B-64694EN-2)	Index table indexing

12.13 SCALING

M

Overview

A programmed figure can be magnified or reduced (scaling).
One scaling method multiples the same magnification rate and the other scaling method (axis-by-axis scaling) multiples the different magnification rate for each axis.
The magnification rate can be specified in the program or by a parameter.

NOTE

To use this function, set bit 5 (SCL) of parameter No.8132 to 1.

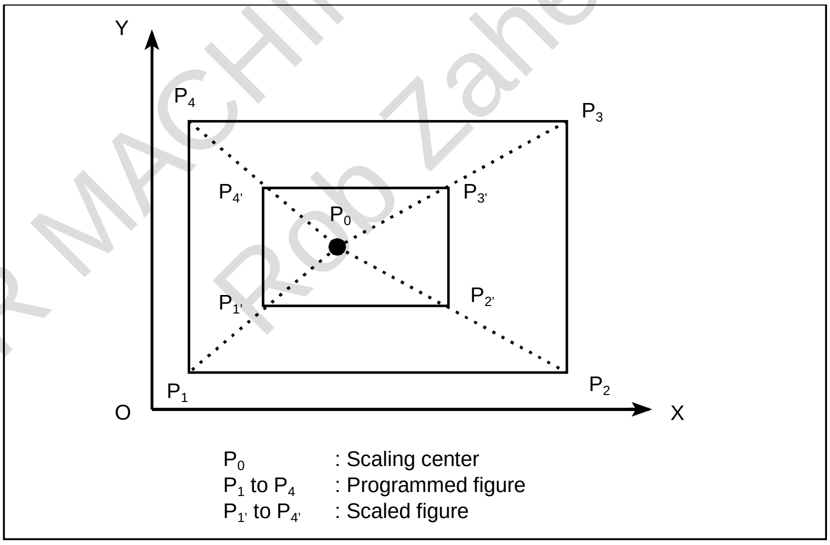


Fig. 12.13 (a) Scaling

Format

Scaling up or down along all axes at the same rate of magnification
(When the bit 6 (XSC) of parameter No. 5400 is set to 0)

Format		Meaning of command
G51 IP_P_ ;	Scaling start	IP_ : Absolute command for center coordinate value of scaling P_ : Scaling magnification
: **:**	Scaling is effective. (Scaling mode)	
G50 ;	Scaling cancel	

Scaling along each axes at a different rate of magnification (mirror image) (When the bit 6 (XSC) of parameter No. 5400 is set to 1)

Format	Meaning of command
G51 IP_ I_J_K_ ; Scaling start : } Scaling is effective. : } (Scaling mode) **G50 ;** Scaling cancel	IP_ : Absolute command for center coordinate value of scaling I_J_K_ : Scaling magnification for basic 3 axes (X, Y, and Z axes) respectively

> ⚠ **CAUTION**
> 1 Specify G51 in a separate block.
> 2 After the figure is enlarged or reduced, specify G50 to cancel the scaling mode.

> **NOTE**
> 1 Entering pocket calculator decimal point input mode (bit 0 (DPI) of parameter No. 3401 = 1) does not cause the units of the magnification rates P, I, J, and K to change.
> 2 Setting the least input increment equal to 10 times the least command increment (bit 7 (IPR) of parameter No. 1004 = 1) does not cause the units of the magnification rates P, I, J, and K to change.
> 3 An attempt to specify 0 as a magnification rate causes alarm PS0142, “ILLEGAL SCALE RATE”, to occur in a G51 block.

Explanation

- Axis for which scaling is to be enabled

For the axis for which scaling is to be enabled, set bit 0 (SCL) of parameter No. 5401 to 1.

- Minimum unit of scaling magnification

Least input increment of scaling magnification is: 0.001 or 0.00001.
It is 0.00001 (one hundred thousandth) if bit 7 (SCR) of parameter No. 5400 is 0 and 0.001 if it is 1.

- Scaling center

Even in incremental command (G91) mode, the scaling center coordinates IP_ specified in the G51 block are assumed those of an absolute position.
If the scaling center coordinates are omitted, the position assumed when G51 is specified is assumed the scaling center.

> ⚠ **CAUTION**
> With the move command subsequent to the G51 block, execute an absolute (G90 mode) position command.
> If no absolute position command is executed after the G51 block, the position when G51 is specified is assumed the scaling center. Once an absolute position command is executed, after that block, the scaling center is assumed the coordinates specified in the G51 block.

- Scaling along each axis at the same rate of magnification

Set bit 6 (XSC) of parameter No. 5400 to 0.
If the scaling magnification P is not specified, the magnification set in parameter No. 5411 is used.
Decimal point input is not accepted as the magnification P. If decimal point input is made, alarm PS0007, “ILLEGAL USE OF DECIMAL POINT”, will occur.
A negative value cannot be specified as the magnification P. If a negative value is specified, alarm PS0006, “ILLEGAL USE OF MINUS SIGN”, will occur.
The allowable magnification range is from 0.00001 to 9999.99999.

- Scaling of each axis, programmable mirror image (negative magnification)

Each axis can be scaled by different magnifications. Also when a negative magnification is specified, a mirror image is applied. The axis subject to the mirror image is the one that contains the scaling center.
Set a bit 6 (XSC) of parameter No. 5400 to 1 to validate each axis scaling (mirror image).
Using I, J, and K, specify the scaling magnifications for the basic 3 axes (X to Z axes). Use parameter No. 1022 to specify which axes to use as the basic 3 axes.
For those of the X to Z axes for which I, J, and K are not specified and for axes other than the basic 3 axes, the magnification set with parameter No. 5421 is used.
A value other than 0 must be set to parameter No. 5421.
Decimal point programming can not be used to specify the rate of magnification (I, J, K).
Magnification can be set within the range ±0.00001 ±9999.99999.

> ⚠ **CAUTION**
>
> Specifying the following commands at the same time causes them to be executed in the order indicated below:
>
> <1> Programmable mirror image (G51.1)
> <2> Scaling (G51) (including a mirror image with a negative magnification)
> <3> Mirror image due to the external switch of the CNC or the settings of the CNC
>
> In this case, the programmable mirror image is effective to the scaling center and magnification as well.
> To specify G51.1 and G51 at the same time, specify them in this order; to cancel them, specify them in the reverse order.

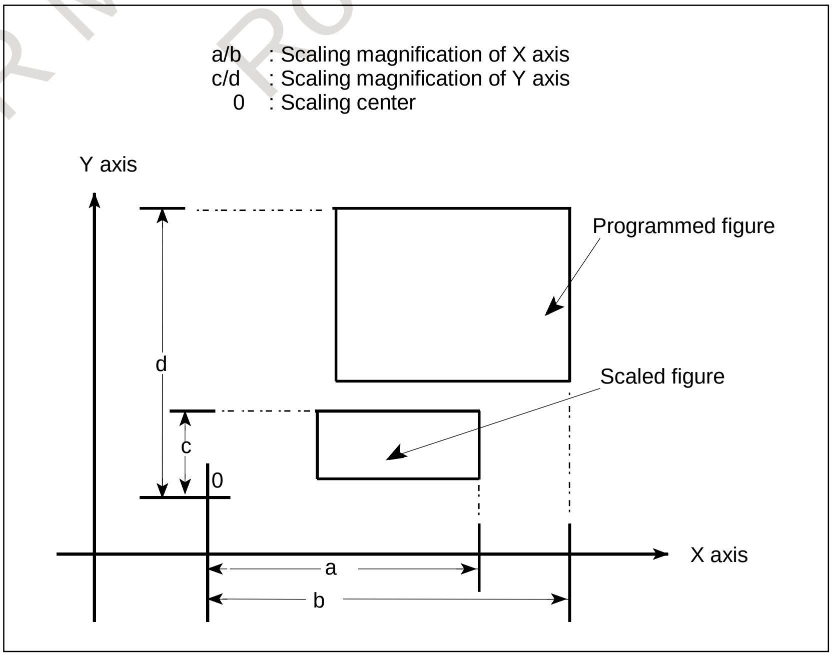


Fig. 12.13 (b) Scaling of each axis

- Scaling of circular interpolation

Even if different magnifications are applied to each axis in circular interpolation, the tool will not trace an ellipse.

G90 G00 X0.0 Y100.0 Z0.0;
G51 X0.0 Y0.0 Z0.0 I2000 J1000; (A magnification of 2 is applied to the X direction and a magnification of 1 is applied to the Y direction.)
G02 X100.0 Y0.0 I0 J–100.0 F500;

Above commands are equivalent to the following command:

G90 G00 X0.0 Y100.0 Z0.0;
G02 X200.0 Y0.0 I0 J–100.0 F500;
(Because the end point is not on an arc, spiral interpolation is assumed.)

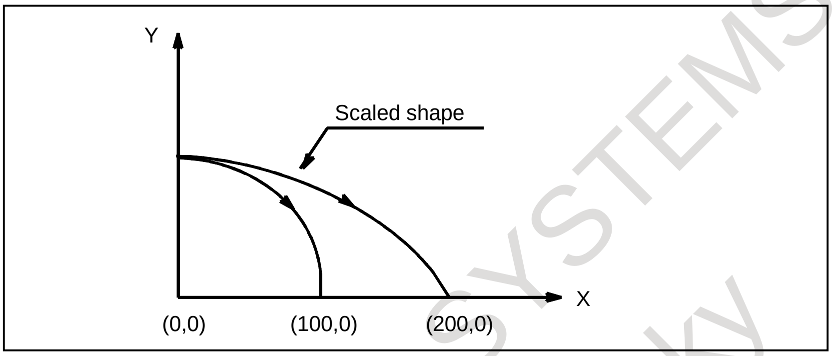


Fig. 12.13 (c) Scaling for circular interpolation1

Even for an R-specified arc, scaling is applied to each of I, J, and K after the radius value (R) is converted into a vector in the center direction of each axis.
If, therefore, the above G02 block contains the following R-specified arc, the operation will be same as that in which I and J are specified.

G02 X100.0 Y0.0 R100.0 F500 ;

- Scaling and coordinate system rotation

If both scaling and coordinate system rotation are specified at the same time, scaling is performed first, followed by coordinate system rotation. In this case, scaling is effective to the rotation center as well.
To specify both of them, specify scaling first and then coordinate system rotation. To cancel them, specify them in the reverse order.

Example

Main program
O1
G90 G00 X20.0 Y10.0 ;
M98 P1000 ;
G51 X20.0 Y10.0 I3000 J2000 ; (× 3 in the X direction and × 2 in the Y direction)
M98 P1000 ;
G17 G68 X35.0 Y20.0 R30.0 ;
M98 P1000 ;
G69 ;
G50 ;
M30 ;

Subprogram
O1000 ;
G01 X20.0 Y10.0 F500 ;
G01 X50.0 ;
G01 Y30.0 ;

```
G01 X20.0 ;
G01 Y10.0 ;
M99 ;
```

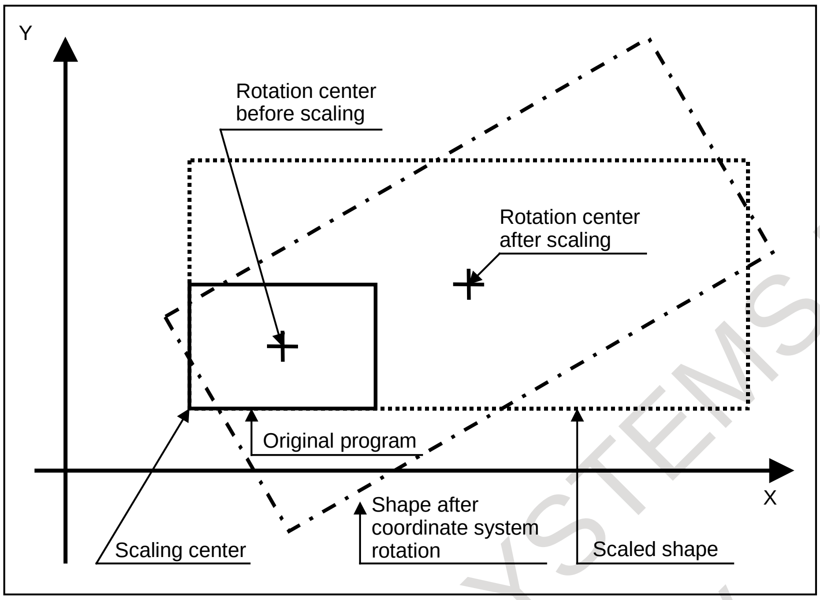


Fig. 12.13 (d) Scaling and coordinate system rotation

- Scaling and optional-angle chamfering/corner rounding

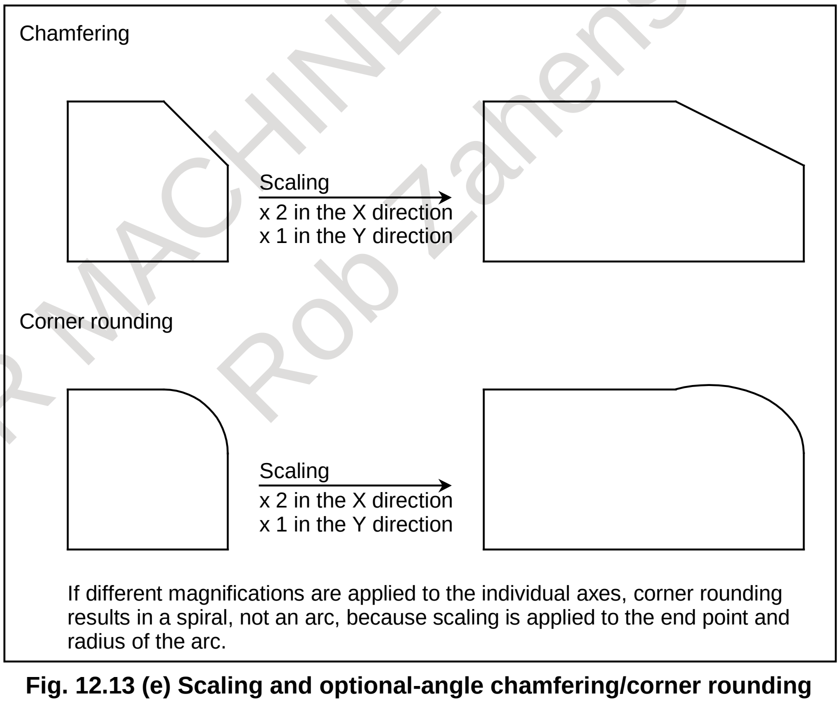


Fig. 12.13 (e) Scaling and optional-angle chamfering/corner rounding

Limitation

- Tool compensation

This scaling is not applicable to cutter/tool nose radius compensation values, tool length offset values, and tool offset values (Fig. 12.13 (f)).

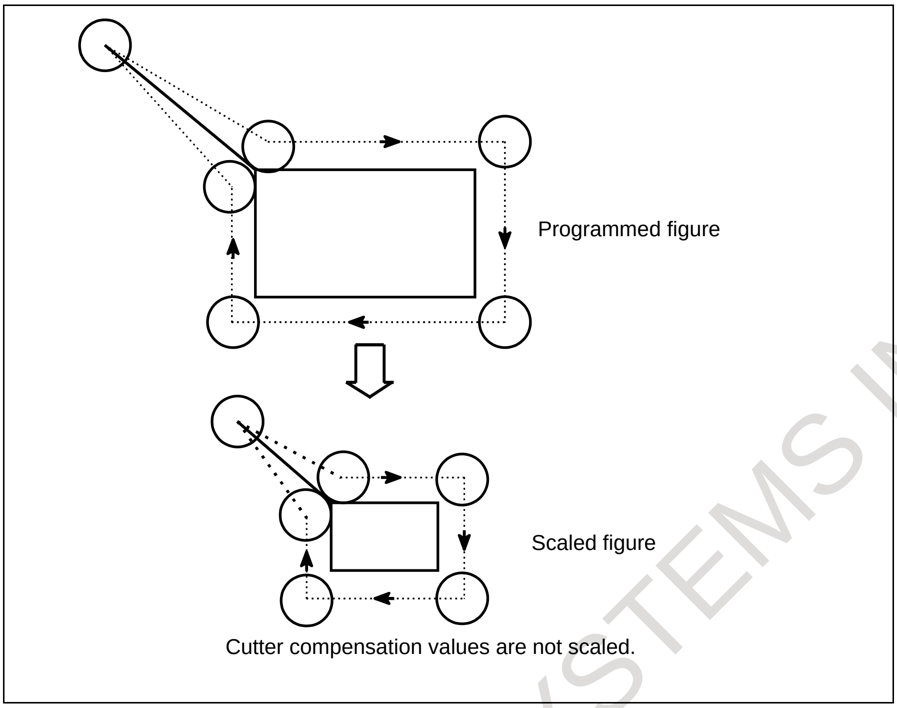


Fig. 12.13 (f) Scaling during cutter compensation

- Invalid scaling

Scaling is not applicable to the Z-axis movement in case of the following canned cycle.

- Cut-in value Q and retraction value d of peck drilling cycle (G83, G73).
- Fine boring cycle (G76)
- Shift value Q of X and Y axes in back boring cycle (G87).

Caution

⚠ **CAUTION**

1 If a parameter setting value is employed as a scaling magnification without specifying P, the setting value at G51 command time is employed as the scaling magnification, and a change of this value, if any, is not effective.
2 Before specifying the G code for reference position return (G27, G28, G29, G30) or coordinate system setting (G52 to G59, G92), cancel the scaling mode. If any of them is specified during scaling, alarm PS0412, “AN UNUSABLE G CODE WAS USED”, will occur.
3 If scaling results are rounded by counting fractions of 5 and over as a unit and disregarding the rest, the move amount may become zero. In this case, the block is regarded as a no movement block, and therefore, it may affect the tool movement by cutter compensation. See the description of cutter compensation.
4 Refrain from scaling on a rotation axis for which the rollover function is enabled. Otherwise, the tool may rotate in a short-cut manner, possibly resulting in unexpected movement.

Note

NOTE

1 The position display represents the coordinate value after scaling.
2 When a mirror image was applied to one axis of the specified plane, the following results:

(1) Circular command	Direction of rotation is reversed.
(2) Cutter/tool nose radius compensation	Offset direction is reversed.
(3) Coordinate system rotation	Rotation angle is reversed.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5400	SCR	XSC						

[Input type] Parameter input
[Data type] Bit path

#6 XSC The setting of a scaling magnification (axis-by-axis scaling) is:
0: Disabled.
1: Enabled.

#7 SCR Scaling (G51) magnification unit:
0: 0.00001 times (1/100,000)
1: 0.001 times

	#7	#6	#5	#4	#3	#2	#1	#0
5401								SCLx

[Input type] Parameter input
[Data type] Bit axis

#0 SCLx Scaling on this axis:
0: Invalidated
1: Validated

5411	Scaling (G51) magnification

[Input type] Setting input
[Data type] 2-word path
[Unit of data] 0.001 or 0.00001 times (Selected using bit 7 (SCR) of parameter No. 5400)
[Valid data range] 1to999999999

This parameter sets a scaling magnification when axis-by-axis scaling is disabled (with bit 6 (XSC) of parameter No. 5400 set to 0). If no scaling magnification (P) is specified in the program, the setting of this parameter is used as a scaling magnification.

NOTE
When bit 7 (SCR) of parameter No. 5400 is set to 1, the valid data range is 1 to 9999999.

5421	Scaling magnification for each axis

[Input type] Setting input
[Data type] 2-word axis
[Unit of data] 0.001 or 0.00001 times (Selected using bit 7 (SCR) of parameter No. 5400)

[Valid data range] -999999999 to -1, 1 to 999999999

This parameter sets a scaling magnification for each axis when axis-by-axis scaling is enabled (with bit 6 (XSC) of parameter No. 5400 set to 1). For the first spindle to the third spindle (X-axis to Z-axis), the setting of this parameter is used as a scaling magnification if scaling magnifications (I, J, K) are not specified in the program.

NOTE
When bit 7 (SCR) of parameter No. 5400 is set to 1, the valid data ranges are -9999999 to -1 and 1 to 9999999.

Alarm and message

- Alarms common to lathe systems and machining center systems

Number	Message	Description
PS0006	ILLEGAL USE OF MINUS SIGN	A minus sign (–) was specified at an NC instruction word or system variable where no minus signal may be specified.
PS0007	ILLEGAL USE OF DECIMAL POINT	A decimal point (.) was specified at an address where no decimal point may be specified, or two or more decimal points were specified.
PS0142	ILLEGAL SCALE RATE	The scaling rate is 0 times or 10000 times or more. Modify the setting of the scaling rate. (G51P_ . . . or G51I_J_K_ . . . or parameter No. 5411 or 5421)
PS0412	ILLEGAL G CODE	During scaling, the G code for reference position return (G27, G28, G29, G30) or coordinate system setting (G52 to G59, G92), is specified. Before specifying any of them, cancel the scaling mode.
PS5007	TOO LARGE DISTANCE	Due to compensation, point of intersection calculation, interpolation or similar reasons, a movement distance that exceeds the maximum permissible distance was specified. Check the programmed coordinates or compensation amounts.

- Alarms only for lathe systems

Number	Message	Description
PS0300	ILLEGAL COMMAND IN SCALING	An illegal G code was specified during scaling. Modify the program.
PS0509	TOOL OFFSET COMMAND IS NOT AVAILABLE	- Tool offset (for the lathe system) was specified in the thread cutting block. - Tool offset was specified (lathe system) in the coordinate system rotation mode, or programmable mirror image mode.

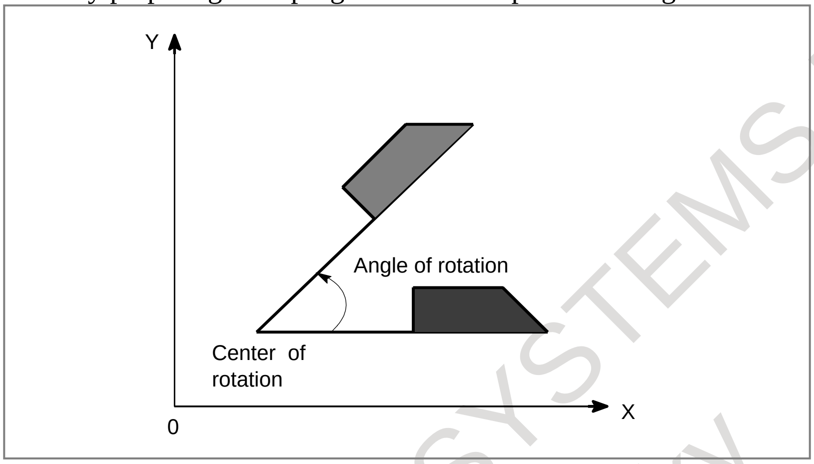
Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Scaling

12.14 COORDINATE SYSTEM ROTATION

Overview

A programmed shape can be rotated. By using this function, it becomes possible, for example, to correct the position of a mounted workpiece using a rotation command when the workpiece is placed with some angle rotated from the machine coordinates. Further, when there is a pattern comprising some identical shapes in the positions rotated from a shape, the time required for programming and the length of the program can be reduced by preparing a subprogram of the shape and calling it after rotation.



Parameter

- Angle specification method of coordinate system rotation

	#7	#6	#5	#4	#3	#2	#1	#0
5400								RIN

[Input type] Parameter input
[Data type] Bit path

#0 RIN Coordinate rotation angle command (R) :
0: Specified by an absolute method
1: Specified by an absolute method (G90) or incremental method (G91)

> **NOTE**
> When G code system A of the lathe system is used, it is specified by an absolute method regardless of the setting of this parameter.

- Angular displacement used when no angular displacement is specified for coordinate system rotation

5410	Angular displacement used when no angular displacement is specified for coordinate system rotation

[Input type] Setting input
[Data type] 2-word path
[Unit of data] 0.001 deg
[Valid data range] -360000 to 360000

This parameter sets the angular displacement for coordinate system rotation. When the angular displacement for coordinate system rotation is not specified with address R in the block where G68 is specified, the setting of this parameter is used as the angular displacement for coordinate system rotation.

NOTE

Write this parameter in the rotation cancel (G69) state when it is written by using the applications (such as PMC window, FOCAS2, and C language executor) or the programmable parameter input (G10).
Even if this parameter is written in the rotation mode, it is not immediately enabled.
When the rotation start (G68) that doesn't include the rotation angle (R_) is specified again, it is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
11600			AX1					

[Input type] Parameter input
[Data type] Bit path

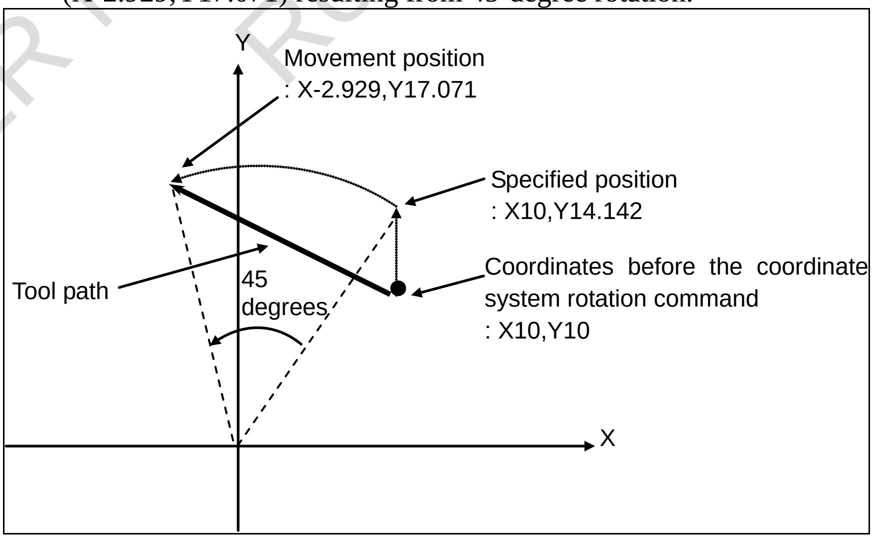
#5 AX1 If, in coordinate system rotation mode, a 1-axis command is issued in absolute mode,

0: First, the specified position is calculated in the coordinate system before rotation, and then the coordinate system is rotated.
1: First, the coordinate system is rotated, and then the tool moves to the specified position in the coordinate system.
(FS16*i*/18*i*/21*i* compatible specification)

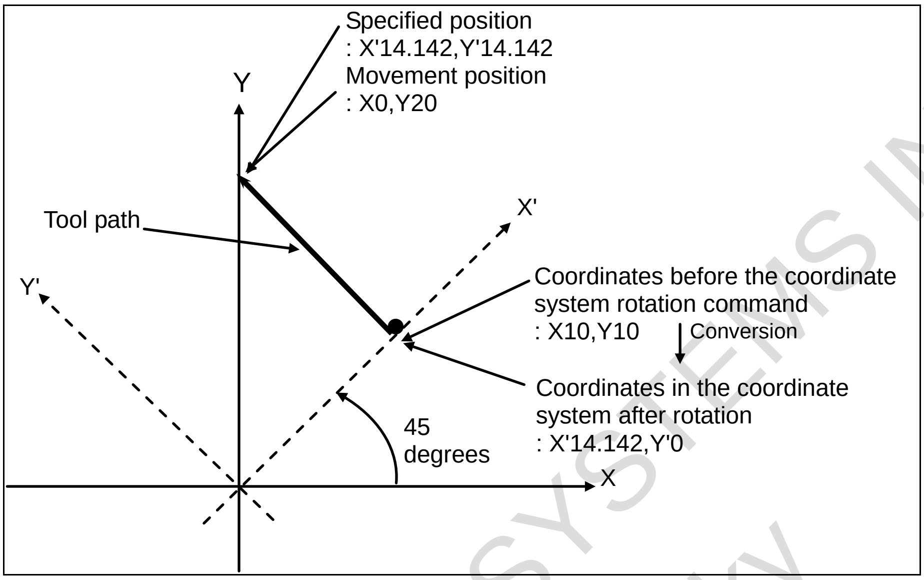
[Example] G90 G0 X0 Y0
G01 X10. Y10. F6000
G68 X0 Y0 R45. ....Coordinate system rotation command
Y14.142 ....1-axis command (1)
G69

When the bit 5 (AX1) of parameter No. 11600 is set to 0:

In the coordinate system (XY) before rotation, the specified position is calculated, and then the coordinate system is rotated. Thus, for the command in (1), the position on the X-axis that is not specified is assumed to be X10, so that the specified position is (X10,Y14.142). Then, the tool moves to the movement position (X-2.929,Y17.071) resulting from 45-degree rotation.



When the bit 5 (AX1) of parameter No. 11600 is set to 1:
The command in (1) converts the coordinates before the coordinate system rotation command, (X10,Y10), into the coordinates (X'14.142,Y'0) in the coordinate system rotated by 45 degrees (X'Y'). Then, the tool moves to the specified position (X'14.142,Y'14.142), i.e., the movement position (X0,Y20).



	#7	#6	#5	#4	#3	#2	#1	#0
11630								FRD

[Input type] Parameter input
[Data type] Bit path

#0 FRD The minimum command unit of the rotation angles of coordinate rotation and 3-dimensional coordinate system conversion is:
0: 0.001 degree.
1: 0.00001 degree. (1/100,000)

Alarm and message

Number	Message	Description
PS0144	ILLEGAL PLANE SELECTED	The coordinate rotation plane and arc or tool radius·tool nose radius compensation plane must be the same. Modify the program.
PS0412	ILLEGAL G CODE	During coordinate system rotation, the G code for reference position return (G27, G28, G29, G30) or coordinate system setting (G52 to G59, G92), is specified. Before specifying any of them, cancel the coordinate system rotation mode.
PS0509	TOOL OFFSET COMMAND IS NOT AVAILABLE	- Tool offset (for the lathe system) was specified in the thread cutting block. - Tool offset was specified (lathe system) in the coordinate system rotation mode, or programmable mirror image mode.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Coordinate system rotation

Limitation

- R-specified arc

When an R-specified arc is specified after beginning the coordinate system rotation mode, please specify the command to the start point by the absolute commands in the previous block.

- Specifying during AI high-precision contour control mode or 3-dimensional coordinate system conversion mode

Coordinate system rotation can always be specified for the M series and can be specified for the T series only when the G code system B or C is used.

- Tool offset

T

If tool offset by shifting the coordinate system is specified (by setting bit 2 (LWT) of parameter No. 5002 to 1 or bit 4 (LGT) of parameter No. 5002 to 0) in the coordinate system rotation mode, alarm PS0509, "TOOL OFFSET COMMAND IS NOT AVAILABLE", will occur.
The same alarm will occur if a tool compensation value is modified by setting bit 6 (EVO) of parameter No. 5001.

Caution

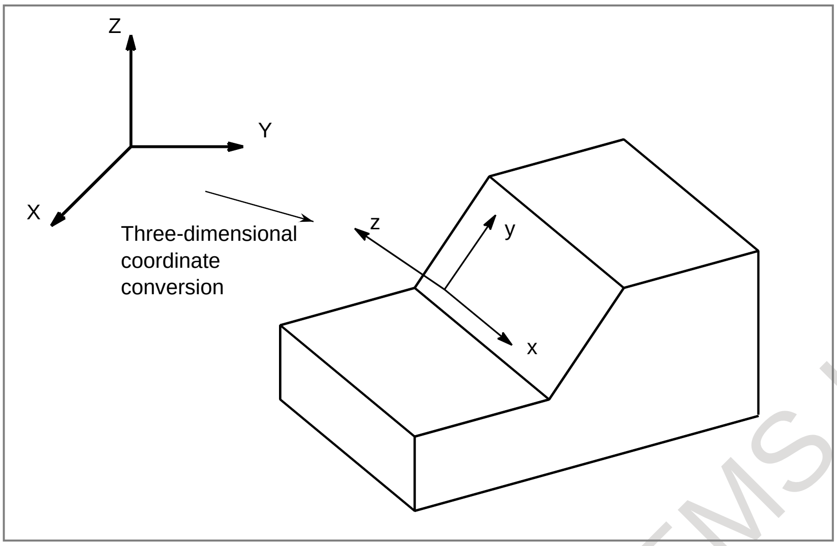
⚠ **CAUTION**

Before specifying the G code for reference position return (G27, G28, G29, G30) or coordinate system setting (G52 to G59, G92), cancel the coordinate system rotation mode. If any of them is specified during coordinate system rotation mode, alarm PS0412, "AN UNUSABLE G CODE WAS USED", will occur.

12.15 3-DIMENSIONAL COORDINATE CONVERSION

Overview

The coordinate system can be rotated about an arbitrary axis by specifying the center of rotation, direction of the axis of rotation, and angular displacement.
This coordinate conversion function is quite useful for 3-dimensional machining on a diesinking machine. By applying this conversion to a program generated for machining on the XY plane, identical machining can be executed on the desired plane.
When rigid tapping is commanded in 3-dimensional coordinate conversion mode, tapping can be done at the angle specified by a 3-dimensional coordinate conversion command. (This is called 3-dimensional rigid tapping.)



Explanation

- Rapid traverse in a canned cycle for drilling

By using bit 1 (D3R) of parameter No. 11221, rapid traverse in the drilling direction in a canned cycle for drilling in the tilted working plane indexing mode or 3-dimensional coordinate conversion mode can be selected and the rapid traverse mode for shift operation in fine boring (G76) or back boring (G87) for the M system can also be selected. When D3R is set to 0, the cutting feed mode is selected; when it is set to 1, the rapid traverse mode is selected. For positioning to the initial level at the start of a canned cycle for drilling, however, the rapid traverse mode is used, regardless of whether D3R is set to 0 or 1. Rapid traverse override can be applied in the rapid traverse mode, and cutting feed override can be applied in the cutting feed mode. D3R can also be used to change the mode of rapid traverse in the rigid tapping direction.

For acceleration/deceleration after interpolation

When bit 1 (D3R) of parameter No. 11221 is set to 1 (for the rapid traverse mode), rapid traverse in the drilling direction in a canned cycle for drilling in the tilted working plane indexing mode or 3-dimensional coordinate conversion mode is accelerated/decelerated along each axis independently according to the time constants specified in parameters Nos. 1620 and 1621. So, when performing positioning of linear interpolation type with D3R set to 1, make the following settings:

Bit 1 of (LRP) of parameter No. 1401=1	Selects positioning of linear interpolation type.
Bit 1 of (PRT) of parameter No. 1603=1	Selects acceleration/deceleration of constant time type.
Parameter No. 1620	Set the same value for all.
Parameter No. 1621	Set the same value for all.

For look-ahead acceleration/deceleration before interpolation

In a canned cycle for drilling, linear rapid traverse in the drilling direction is performed at all times.

Feedrate

When bit 1 (D3R) of parameter No. 11221 is set to 1 (for the rapid traverse mode), the rapid traverse rate in the drilling direction in a canned cycle for drilling in the tilted working plane indexing mode or 3-dimensional coordinate conversion mode is clamped to a parameter value indicated below if the rapid traverse override is 100%. In the rapid traverse override F0 mode, however, the rapid traverse rate is clamped to the value of parameter No. 1421 or a parameter value listed in Table 12.15 (a), whichever smaller.

Table 12.15 (a) Feedrate in the rapid traverse mode

Parameter No. 5412=0	Parameter No. 1420
Parameter No. 5412≠0	Value of parameter No. 1420 or value of parameter 5412, whichever smaller

However, when the external deceleration function is enabled, the feedrate is clamped to a parameter value listed in Table 12.15 (a) that is overridden or an external deceleration speed, whichever smaller.

NOTE
1. Set a rapid traverse rate for each axis in parameter No. 1420 and a tangential rapid traverse rate in parameter No. 5412.
2. If rapid traverse of nonlinear interpolation type is performed, the feedrate is clamped to the value of parameter No. 1420, regardless of the setting of parameter No. 5412.
3. When the cutting feed mode is selected (when bit 1 (D3R) of parameter No. 11221 is set to 0), set a tangential rapid traverse rate in parameter No. 5412.

- Cutting feedrate clamp

The cutting feedrate is clamped so that the feedrate of each real axis after the 3-dimensional coordinate system conversion does not exceed the maximum cutting feedrate（Parameter No.1432 if look-ahead acceleration/deceleration before interpolation is enabled and parameter No.1430 otherwise).

- Interlock signal for each axis direction, interlock signal for each axis, external deceleration signal

When bit 2 (D3IT) of parameter No.11600 is set to 1 and bit 5 (ITM) of parameter No.11223 is set to 1, the following signals are enabled for axis of machine coordinate system.
- Interlock signal for each axis direction during manual intervention / manual handle interrupt in 3-dimensional coordinate system conversion mode
- Interlock signal for each axis during manual intervention / manual handle interrupt in 3-dimensional coordinate system conversion mode
- External deceleration signal during manual intervention / manual handle interrupt in 3-dimensional coordinate system conversion mode
- External deceleration signal during 3-dimensional coordinate conversion

By this setting, all signals of controlled axis type are enabled for axis of machine coordinate system during 3-dimensional coordinate conversion.

Table12.15 (b) Relationship between coordinate system of target axis of each signal during 3-dimensional coordinate conversion (automatic operation) and parameter setting

D3IT	ITM	Interlock for each axis direction	Interlock for each axis	External deceleration
0	-	Programming coordinate system	Interlock all X, Y, Z-axis	Programming coordinate system
1	0	Machine coordinate system	Machine coordinate system	Programming coordinate system
	1	Machine coordinate system	Machine coordinate system	Machine coordinate system

Table12.15 (c) Relationship between coordinate system of target axis of each signal during manual intervention / manual handle interrupt in 3-dimensional coordinate system conversion mode and parameter setting

D3IT	ITM	Interlock for each axis direction	Interlock for each axis	External deceleration
0	-	Programming coordinate system	Programming coordinate system	Programming coordinate system
1	0	Programming coordinate system	Programming coordinate system	Programming coordinate system
	1	Machine coordinate system	Machine coordinate system	Machine coordinate system

- Reset

If a reset occurs during 3-dimensional coordinate system conversion mode, the mode is canceled and the continuous-state G code is changed to G69. Bit 2 (D3R) of parameter No. 5400 determines whether just the G69 code is used to cancel the 3-dimensional coordinate system conversion mode (G68). When this setting is selected, a CNC reset by a reset operation or by an input signal from the PMC will not cancel the 3-dimensional coordinate system conversion mode.

- System variables of skip positions

System variable number and coordinate system of skip are as Table. The coordinate system of #100151- and #151001- changes according to bit 5 (LV3) of parameter No.5400.

Table12.15 (d) Parameter LV3 and coordinate system of skip positions

System variable number	Coordinate system of skip positions	
	Parameter LV3=0	Parameter LV3=1
#100151 -	Workpiece coordinate system	Program coordinate system
#151001 -	Program coordinate system	Workpiece coordinate system
#151101 -	Machine coordinate system	

- Program restart

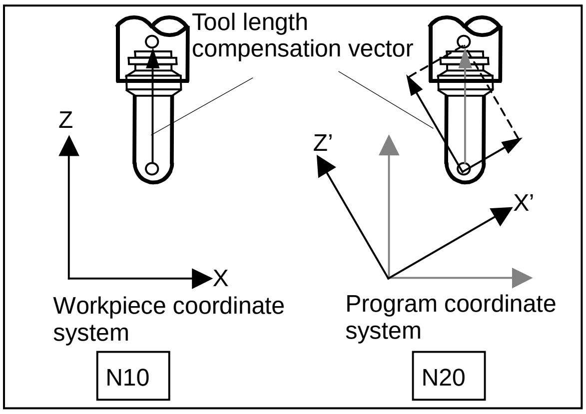
If the block subject to a restart is in 3-dimensional coordinate conversion mode G68 (machining center system)/G68.1 (lathe system), it is possible to select whether the movement of the axes to the restart position should be in the program coordinate system or the workpiece coordinate system, by using bit 1 (3DD) of parameter No.7301. If parameter bit 3DD is 0, the axes move to the restart position one at a time, as seen from the program coordinate system, and if it is 1, they move to the restart position one at a time, as seen from the workpiece coordinate system.
The "DESTINATION" and "DISTANCE TO GO" displays follow the setting of 3DD.
The "ABSOLUTE" display follows the bit 6 (DAK) of parameter No.3106.

- 3-dimensional coordinate system conversion during tool length compensation

By setting bit 3 (TLC) of parameter No. 11221 to 1, 3-dimensional coordinate system conversion can be used during tool length compensation. In this case, the tool length compensation vector is converted to the program coordinate system subject to 3-dimensional coordinate system conversion. This makes machining possible while keeping the tool direction to the Z-axis direction of the workpiece coordinate system.
Tool length compensation and 3-dimensional coordinate system conversion must be nested inside each other.

(Example)

```
----N10 G43 H1 X_ Y_ Z_ ;
        :
  -- N20 G68 X0 Y0 Z0 I0.0 J1.0 K0.0 R-30.0 ;
        :
  -- G69 ;
        :
---- G49 X_ Y_ Z_ ;
```



- Constant surface speed control

Constant surface speed control is exercised by using as the reference the following axis according to bit 6 (3CS) of parameter No.11221. (Refer to the explanation of "CONSTANT SURFACE SPEED CONTROL" for details)

- 3CS=0 : the machine axis specified in address P in a G96 block or the machine axis (in the actually operating workpiece coordinate system) set in parameter No.3770.
- 3CS=1 : the axis in the feature coordinate system

- Angular displacement R

A positive angular displacement R indicates a clockwise rotation along the axis of rotation. Specify angular displacement R in 0.001 degrees within the range of -360000 to 360000.
To specify angular displacement R in 0.00001 degrees (one hundred-thousandth), set bit 0 (FRD) of parameter No.11630 to 1. In this case, angular displacement R is specified within the range of -36000000 to 36000000.

Limitation

- Manual reference position return

An attempt to make a manual reference position return during the 3-dimensional coordinate conversion mode results in alarm PS5324, "REFERENCE RETURN INCOMPLETE", being issued. Before attempting to make a manual reference position return, cancel the 3-dimensional coordinate conversion mode.

- Positioning by optimum accelerations

The function for positioning by optimum accelerations is disabled during the 3-dimensional coordinate system conversion mode.

T

- Relationship between 3-dimensional coordinate system conversion and tool offset

When using a tool offset command, nest the tool offset command within the 3-dimensional coordinate system conversion mode.
(Example)

```
G68.1 X100.0 Y100.0 Z100.0 I0.0 J0.0 K1.0 R45.0 ;
  :
T0101;
  :
T0100;
  :
G69.1 ;
```

When tool geometry and wear compensation with coordinate shift is enabled and it is not canceled by reset, if tool geometry and wear compensation is executed in 3-dimensional coordinate system conversion mode, 3-dimensional coordinate system conversion is not canceled by reset. (Even if it is a setting that 3-dimensional coordinate system conversion is canceled by reset (parameter D3R(No.5400#2)=0), it is not canceled.)

- PMC axis control

If a command related to 3-dimensional coordinate conversion is specified during PMC axis control, the alarm PS5131 is issued.

Signal

3-dimensional coordinate system conversion manual interruption switch signal M3R<Gn031.3>

[Classification] Input signal

[Function] Specifies whether to apply 3-dimensional coordinate system conversion to a manual operation (manual continuous feed, manual incremental feed, or manual handle feed) in the 3-dimensional coordinate conversion mode.

[Operation] When this signal is set to 0, 3-dimensional coordinate system conversion is not applied (the workpiece coordinate system is used).
When this signal is set to 1, 3-dimensional coordinate system conversion is applied (the program coordinate system is used).

3-dimensional coordinate conversion mode signal D3ROT<Fn062.6>

[Classification] Output signal

[Function] Posts whether the 3-dimensional coordinate conversion mode is set.

[Output cond.] When this signal is set to 1, the 3-dimensional coordinate conversion mode is set.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn031					M3R			

	#7	#6	#5	#4	#3	#2	#1	#0
Fn062		D3ROT						

Parameter

- Setting relative position and absolute position

	#7	#6	#5	#4	#3	#2	#1	#0
3104	DAC							
	DAC	DAL		DRL				

[Input type] Parameter input

[Data type] Bit path

#4 DRL Relative position
- 0: The actual position displayed takes into account tool length offset.
- 1: The programmed position displayed does not take into account tool length offset.

#6 DAL Absolute position
0: The actual position displayed takes into account tool length offset.
1: The programmed position displayed does not take into account tool length offset.

> **NOTE**
> In lathe systems, whether to exclude a tool offset when displaying the absolute position is determined by the setting of bit 1 (DAP) of parameter No. 3129.

#7 DAC When an absolute position are displayed:
0: Values not excluding the amount of travel based on cutter compensation and tool nose radius compensation are displayed.
1: Values excluding the amount of travel based on cutter compensation and tool nose radius compensation (programmed positions) are displayed.

> **NOTE**
> When the parameter DAC is set to 1, in a command such as circle interpolation where the cutter compensation vector is changed constantly, an absolute position is not correctly displayed during its interpolation except its start point and its end point.

- Setting absolute coordinates in the 3-dimensional coordinate conversion mode

	#7	#6	#5	#4	#3	#2	#1	#0
3106		DAK						

[Input type] Setting input
[Data type] Bit

#6 DAK Specifies whether to display coordinates in the program coordinate system or workpiece coordinate system as absolute coordinates when the 3-dimensional coordinate conversion mode or the tilted working plane indexing mode is set.
0: Display coordinates in the program coordinate system.
1: Display coordinates in the workpiece coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
5007	3OF	3OC						

[Input type] Parameter input
[Data type] Bit path

#6 3OC If tool length compensation is not cancelled before 3-dimensional coordinate conversion is specified, an alarm is:
0: Not raised.
1: Raised. (Alarm (PS0049) “ILLEGAL COMMAND(G68,G69)”)

#7 3OF If 3-dimensional coordinate conversion is not nested with a command for tool length compensation, or if 3-dimensional coordinate conversion is specified during tool length compensation and another command for tool length compensation is specified:
0: No alarm is issued.
1: The alarm (PS0049) “ILLEGAL COMMAND(G68,G69)” is issued.

Example 1) An alarm is raised in the following cases:

```
1) G68 X_ Y_ Z_ I_ J_ K_ R_ ;
     :
   G43 H1 ;
     :
   G69 ;                          ← PS0049 raised
     :
   G49 ;
2) G43 H1 ;
     :
   G68 X_ Y_ Z_ I_ J_ K_ R_ ;
     :
   G49(G43H2,H2 etc.) ;           ← PS0049 raised
     :
   G69 ;
```

Example 2) No alarm is raised in the following cases:

```
3) G68 X_ Y_ Z_ I_ J_ K_ R_ ;
     :
   G43 H1 ;
     :
   G49 ;
     :
   G69 ;
4) G43 H1 ;
     :
   G68 X_ Y_ Z_ I_ J_ K_ R_ ;
     :
   G69 ;
     :
   G49 ;
```

> **NOTE**
> A command to cancel tool length compensation (G28, etc.) will not cause an alarm to be raised. If such a command is specified in the G68 mode, program as indicated in 3) above.
>
> ```
> G43 H1 ;
>   :
> G68 X_ Y_ Z_ I_ J_ K_ R_ ;
>   :
> G28 X_ Y_ Z_ ;                 ← Offset is cancelled.
>   :                              No alarm is raised.
> G69 ;
> ```


	#7	#6	#5	#4	#3	#2	#1	#0
5400			LV3			D3R		

[Input type] Parameter input
[Data type] Bit path

#2 D3R When Reset is done by reset operation or reset signal from PMC, 3-dimensional coordinate system conversion mode and tilted working plane indexing mode is:
0: Canceled.
1: Not canceled.

#5 LV3 When system variables #100101 to #100132 (current position coordinates) and #100151 to #100182 (skip coordinates) are read in the 3-dimensional coordinate conversion mode or tilted working plane indexing mode:
0: Coordinates of the workpiece coordinate system can be read.
1: Coordinates of the program coordinate system after 3-dimensional coordinate conversion or tilted working plane indexing can be read.
This parameter applies also to system variables #5041 to #5060 (current position coordinates) and #5061 to #5080 (skip coordinates).

5412	Rapid traverse rate for canned cycle for drilling in 3-dimensional coordinate conversion mode

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets a rapid traverse rate for canned cycle for drilling in the 3-dimensional coordinate conversion mode or the tilted working plane indexing mode.

	#7	#6	#5	#4	#3	#2	#1	#0
7301							3DD	

[Input type] Parameter input
[Data type] Bit path

#1 3DD In program restart operation, when the restart block is in 3-dimensional coordinate conversion mode G68 (machining center system) or G68.1 (lathe system), the tool moves to the restart point along each axis:
0: According to the program coordinate system in dry run.
1: According to the workpiece coordinate system in dry run.
The restart coordinates and restart travel distance are also displayed in the coordinate system set in this parameter.

NOTE
The change made to this parameter during program restart operation is ignored.

	#7	#6	#5	#4	#3	#2	#1	#0
11216								D3O

[Input type] Parameter input
[Data type] Bit path

#0 D3O When the bit 1 (D3M) of parameter No.11600 set to “1” and Override is 0%, during Tilted working plane indexing or 3-dimensional coordinate system conversion mode, the axis moving signals “MV1 to MV8<Fn102>” :
- 3-dimensional coordinate system conversion
- Tilted working plane indexing
0: becomes 0.
1: maintains the state of “1”.

NOTE
When the manual operation (Jog feed / Incremental feed / Manual handle interrupt)is being executed, the bit 0 (D3O) of parameter No.11216 become invalid. It becomes operation corresponding to D3O=0.

	#7	#6	#5	#4	#3	#2	#1	#0
11221		3CS			TLC	3DW	D3R	

[Input type] Parameter input
[Data type] Bit path

#1 D3R In the 3-dimensional coordinate system conversion mode or tilted working plane indexing mode, rapid traverse in canned cycle for drilling is:
0: Performed in the cutting feed mode.
1: Performed in the rapid traverse mode.

#2 3DW If, in the 3-dimensional coordinate system conversion mode, workpiece coordinate system selection using a G code is specified, the selection:
0: Operates in accordance with conventional specifications. (The workpiece coordinate system difference is reflected in the program coordinate system direction.)
1: Operates in accordance with the same specifications as those of workpiece coordinate system selection (bit 6 (3TW) of parameter No. 1205 = 1) during the tilted working plane indexing command. (The workpiece coordinate system difference is reflected in the workpiece coordinate system direction.)

⚠ **CAUTION**
If this parameter is set to 1, only G54 to G59 and G54.1 can be specified. If G52 and G92 are specified, alarm PS5462, "ILLEGAL COMMAND (G68.2/G69)" is issued.
If G54 to G59 and G54.1 are specified, buffering is suppressed.

#3 TLC During tool length compensation, 3-dimensional coordinate conversion:
0: Cannot be used.
1: Can be used.

#6 3CS The spindle speed calculation of constant surface speed control during the 3-dimensional coordinate system conversion / the tilted working plane indexing is :
0: Based on the workpiece coordinate system.
1: Based on the program coordinate system (3-dimensional coordinate system conversion) / the feature coordinate system (Tilted working plane indexing).

	#7	#6	#5	#4	#3	#2	#1	#0
11223			ITM					

[Input type] Parameter input
[Data type] Bit path

#5 ITM When bit 2 (D3IT) of parameter No.11600 = 1, during 3-dimensional coordinate conversion, the following signals are:
0: Enabled for axis of programming coordinate system.
1: Enabled for axis of machine coordinate system.
(Intended signals)

- Interlock signal for each axis direction during manual intervention / manual handle interrupt in 3-dimensional coordinate system conversion mode
- Interlock signal for each axis during manual intervention / manual handle interrupt in 3-dimensional coordinate system conversion mode
- External deceleration signal during manual intervention / manual handle interrupt in 3-dimensional coordinate system conversion mode
- External deceleration signal during 3-dimensional coordinate conversion

	#7	#6	#5	#4	#3	#2	#1	#0
11600				D3A		D3IT	D3MV	

[Input type] Parameter input
[Data type] Bit path

#1 D3MV In following modes, axis moving signals MV1 to MV8 <Fn102> is:
- 3-dimensional coordinate system conversion
- Tilted working plane indexing

0: The signals for axes on programming coordinate system.
1: The signals for axes on workpiece coordinate system.

#2 D3IT In the 3-dimensional coordinate system conversion mode, the valid interlock signals (interlock signal for each axis (*ITx) or interlock signal for each axis direction (MITx, PITx)) are:

0: The signals for all of the target axes for 3-dimensional coordinate system conversion.
1: The signals for axes along which a movement is made during 3-dimensional coordinate system conversion.

(Example)
In case of this parameter is set to 0:
When interlock signal for each axis (X-axis) is enabled, all axes for 3-dimensional coordinate system conversion are interlocked.
When interlock signal for each axis direction (+X-axis) is enabled, the axis is interlocked in +X direction of programming coordinate system of 3-dimensional coordinate system conversion.
In case of this parameter is set to 1:
When interlock signal for each axis (X-axis) is enabled, the X axis is interlocked in machine coordinate system.

#4 D3A In 3-dimensional coordinate system conversion cancellation, if the compensation vector has not been canceled:

0: Alarm PS5462, "ILLEGAL COMMAND (G68.2/G69)", is issued.
1: No alarm is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
11630								FRD

[Input type] Parameter input
[Data type] Bit path

#0 FRD The minimum command unit of the rotation angles of coordinate rotation and 3-dimensional coordinate system conversion is:

0: 0.001 degree.
1: 0.00001 degree. (1/100,000)

	#7	#6	#5	#4	#3	#2	#1	#0
19602			D3D					

[Input type] Parameter input
[Data type] Bit path

#5 D3D Specifies whether to display the distance to go in the program coordinate system or workpiece coordinate system during the 3-dimensional coordinate system conversion or the tilted working plane indexing.
0: Display the distance to go in the program coordinate system.
1: Display the distance to go in the workpiece coordinate system.

Alarm and message

Number	Message	Description
PS0047	ILLEGAL AXIS SELECT	Two or more parallel axes (in parallel with a basic axis) have been specified upon start-up of 3-dimensional coordinate conversion.
PS0048	BASIC 3 AXIS NOT FOUND	Start-up of 3-dimensional coordinate conversion has been attempted, but the three basic axes used when Xp, Yp, or Zp is omitted are not set in parameter No. 1022.
PS5043	TOO MANY G68 NESTING	3-dimensional coordinate conversion has been specified three or more times. To perform another coordinate conversion, perform cancellation, then specify the coordinate conversion.
PS5044	G68 FORMAT ERROR	Errors for 3-dimensional coordinate conversion command are: (1) No I, J, or K command was issued in 3-dimensional coordinate conversion command block. (2) All of I, J, or K command were 0 in 3-dimensional coordinate conversion command block. (3) No rotation angle R was commanded in 3-dimensional coordinate conversion command block.
PS5131	NC COMMAND IS NOT COMPATIBLE	The PMC axis control and 3-dimensional coordinate conversion or polar coordinate interpolation were specified simultaneously. Modify the program or ladder.
PS5324	REFERENCE RETURN INCOMPLETE	No manual reference position return can be made during 3-dimensional coordinate conversion or tilted working plane indexing execution.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	3-dimensional coordinate conversion

12.16 TILTED WORKING PLANE INDEXING

12.16.1 Tilted Working Plane Indexing

Overview

Programming for creating holes, pockets, and other figures in a datum plane tilted with respect to the workpiece would be easy if commands can be specified in a coordinate system fixed to this plane (called a feature coordinate system). This function enables commands to be specified in the feature coordinate system. The feature coordinate system is defined in the workpiece coordinate system.
For explanations about the relationship between the feature coordinate system and workpiece coordinate system, see Fig. 12.16.1 (a).

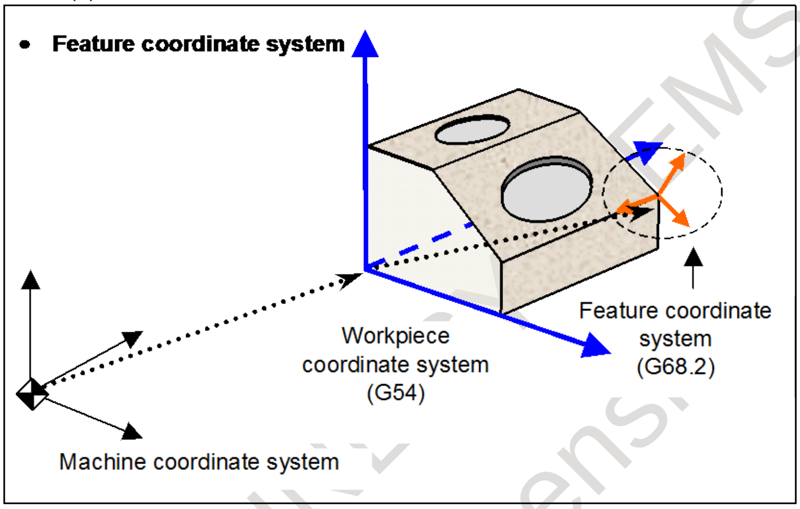


Fig. 12.16.1 (a) Feature coordinate system

The G68.2 command causes the programming coordinate system to switch to the feature coordinate system. The commands in all subsequent blocks are assumed to be specified in the feature coordinate system until G69 appears.
If G68.2 specifies the relationship between the feature coordinate system and the workpiece coordinate system, G53.1 automatically specifies the +Z direction of the feature coordinate system as the tool axis direction even if no angle is specified for the rotation axis. (See Fig. 12.16.1 (c).)
For explanations about the tool axis direction, see Fig. 12.16.1 (b).

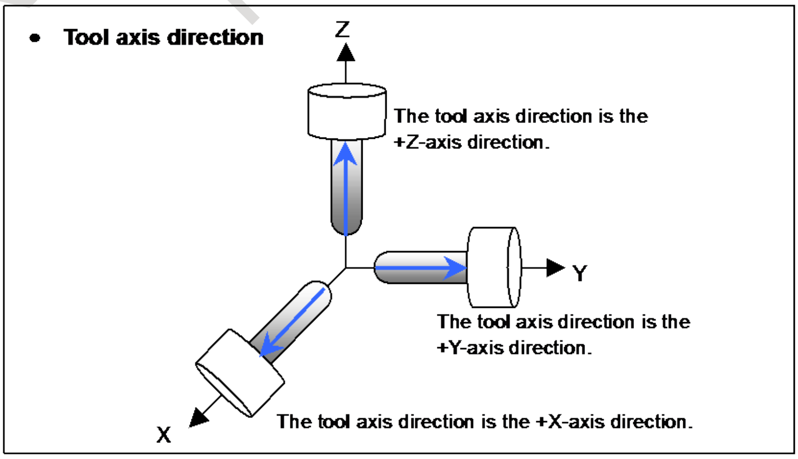


Fig. 12.16.1 (b) Tool axis direction

This function regards the direction normal to the machining plane as the +Z-axis direction of the feature coordinate system. After the G53.1 command, the tool is controlled so that it remains perpendicular to the machining plane.

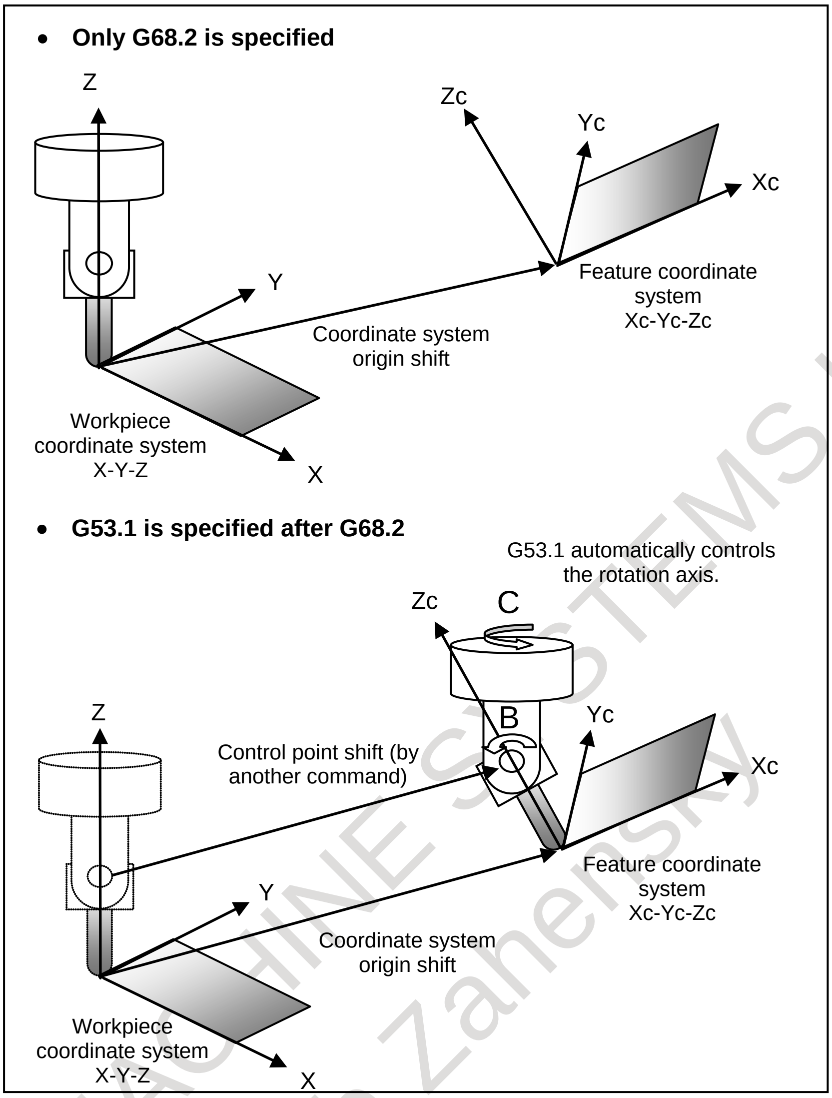


Fig. 12.16.1 (c) G68.2 and G53.1 commands

This function is applicable to the following machine configurations. (See Fig. 12.16.1 (d).)
<1> Tool rotation type machine controlled with two tool rotation axes
<2> Table rotation type machine controlled with two table rotation axes
<3> Composite type machine controlled with one tool rotation axis and one table rotation axis

The function can also be used for a machine configuration in which the rotation axis for controlling the tool does not intersect the rotation axis for controlling the table.

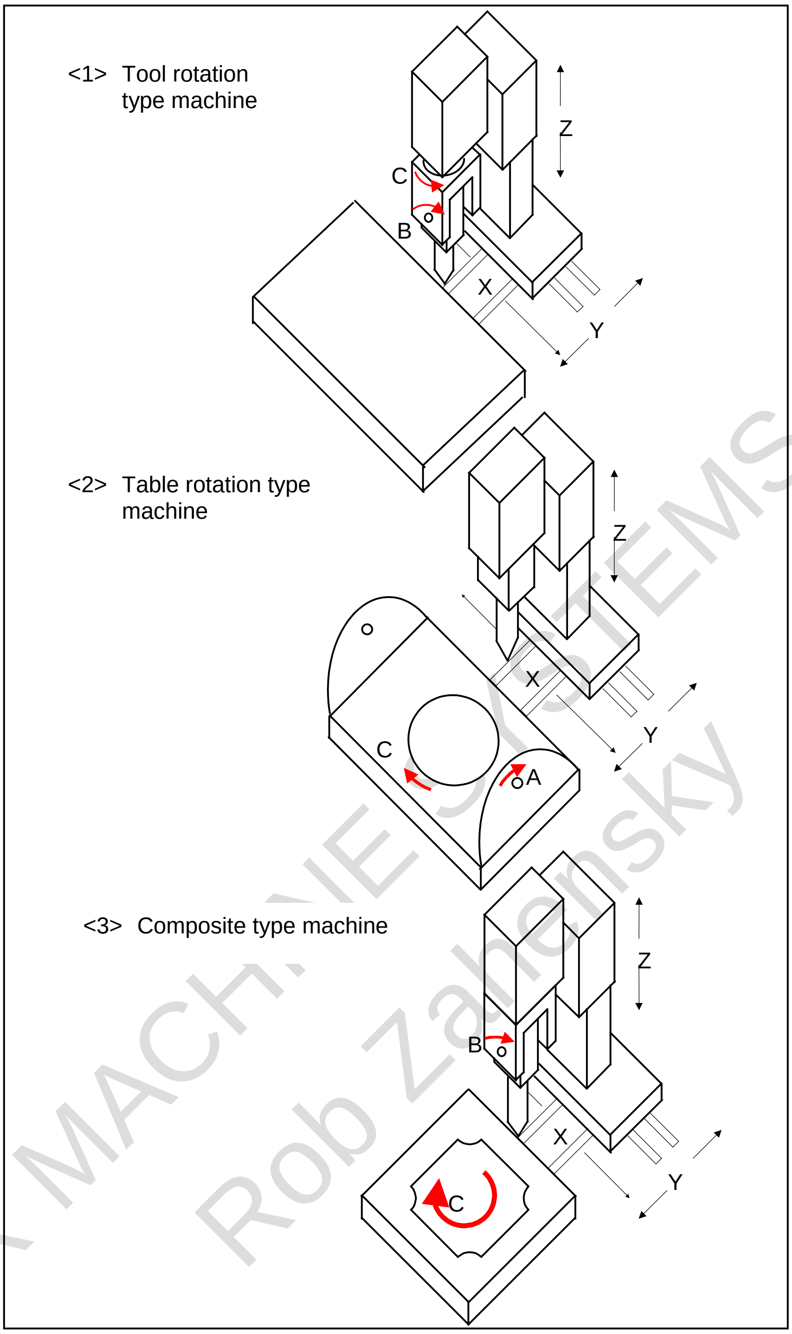


Fig. 12.16.1 (d) Three types of 5-axis machine

12.16.1.1 Tilted working plane indexing based on Eulerian angle

- Coordinate conversion using an Euler's angle

Coordinate conversion by rotation is assumed to be performed around the workpiece coordinate system origin.

Let the coordinate system obtained by rotating the workpiece coordinate system around the Z-axis by an angle of α degrees be coordinate system 1. Similarly, let the coordinate system obtained by rotating coordinate system 1 around the X'-axis by an angle of β be coordinate system 2. The feature coordinate system is the coordinate system obtained by shifting the coordinate system that is obtained by rotating coordinate system 2 around the Z"-axis through an angle of γ degrees from the workpiece coordinate system origin by (Xo, Yo, Zo).

Fig. shows the relationship between the workpiece coordinate system and the feature coordinate system. The figure also gives examples of displacement on the X-Y plane.

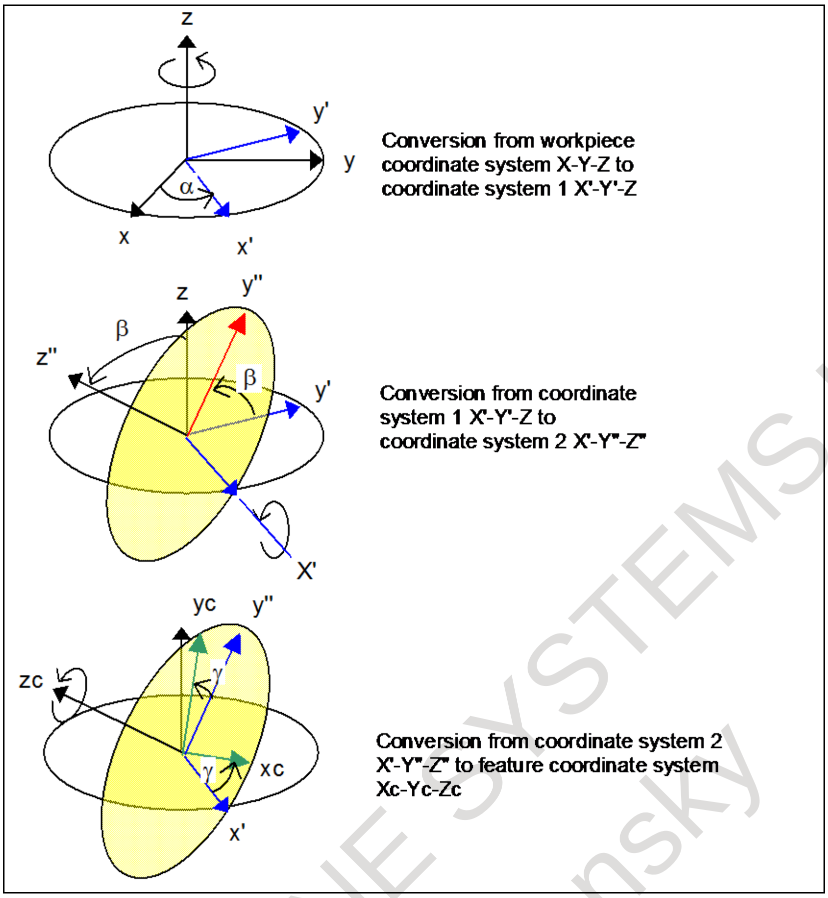


Fig. 12.16.1.1 (a) Coordinate conversion using an Euler's angle

- I0 J0 K0 command

When I0 J0 K0 is specified as an Euler's angle, the alarm PS5457, "G68.2/G68.3 FORMAT ERROR" usually occurs. When bit 1 (ATW) of parameter No. 13451 is set to 1, the feature coordinate system with a tilted angle of 0 degree is used.

12.16.1.2 General specifications of the tilted working plane indexing

- Constant surface speed control

Constant surface speed control is exercised by using as the reference the following axis according to bit 6 (3CS) of parameter No.11221. (Refer to the explanation of "CONSTANT SURFACE SPEED CONTROL" for details)

- 3CS=0 : the machine axis specified in address P in a G96 block or the machine axis (in the actually operating workpiece coordinate system) set in parameter No. 3770.
- 3CS=1 : the axis in the feature coordinate system

- Rapid traverse in a canned cycle for drilling

By using bit 1 (D3R) of parameter No. 11221, rapid traverse in the drilling direction in a canned cycle for drilling during the tilted working plane indexing mode, 3-dimensional coordinate system conversion mode can be selected and the rapid traverse mode for shift operation in fine boring (G76) or back boring (G87) for the M system can also be selected. When parameter D3R is set to 0, the cutting feed mode is selected. When parameter D3R is set to 1, the rapid traverse mode is selected. For positioning to the initial level at the start of a canned cycle for drilling, however, the rapid traverse mode is used, regardless of the setting of parameter D3R. Rapid traverse override is applied during the rapid traverse mode, and cutting feed override is applied during the cutting feed mode. Parameter D3R is also usable to change the mode of rapid traverse in the rigid tapping direction.

For acceleration/deceleration after interpolation

When parameter D3R is set to 1 (for the rapid traverse mode), rapid traverse in the drilling direction in a canned cycle for drilling during the tilted working plane indexing mode, 3-dimensional coordinate system conversion mode is accelerated/decelerated on each axis independently according to the time constants specified in parameter No. 1620 and parameter No. 1621. So, when performing positioning of linear interpolation type with parameter D3R set to 1, make the following settings:

Bit 1 (LRP) of parameter No. 1401=1		Selects positioning of linear interpolation type.
Bit 4 (PRT) of parameter No. 1603=1		Selects acceleration/deceleration of constant time type.
Parameter No. 1620	(Tab)	Set the same value for all.
Parameter No. 1621	(Tab)	Set the same value for all.

For look-ahead acceleration/deceleration before interpolation

In a canned cycle for drilling, linear rapid traverse in the drilling direction is performed at all times.

Feedrate

When parameter D3R is set to 1 (for the rapid traverse mode), the rapid traverse rate in the drilling direction in a canned cycle for drilling during the tilted working plane indexing mode, 3-dimensional coordinate system conversion mode is clamped to a parameter value indicated below if the rapid traverse override is 100%. In the rapid traverse override F0 mode, however, the rapid traverse rate is clamped to the value of parameter No. 1421 or a parameter value in Table , whichever smaller.

Table 12.16.1.2 (a) Feedrate for rapid traverse mode

Parameter No. 5412=0	Parameter No. 1420
Parameter No. 5412≠0	Value of parameter No. 1420 or value of parameter No. 5412, whichever smaller

However, when the external deceleration function is enabled, the feedrate is clamped to a parameter value in Table that is overridden or an external deceleration speed, whichever smaller.

NOTE

1. Set a rapid traverse rate for each axis in parameter No. 1420, and set a tangential rapid traverse rate in parameter No. 5412.
2. If rapid traverse of nonlinear interpolation type is performed, the feedrate is clamped to the value of parameter No. 1420, regardless of the setting of parameter No. 5412.
3. When the cutting feed mode is selected (parameter D3R = 0), set a tangential rapid traverse rate in parameter No. 5412.

- Workpiece coordinate system selection command during the tilted working plane indexing

By executing the workpiece coordinate system selection command (G54 to G59, G54.1) during the tilted working plane indexing when bit 6 (3TW) of parameter No. 1205 is 1, it is possible to change the workpiece coordinate system. In this case, the coordinate system zero point shift of the tilted working plane indexing is maintained.

If an attempt is made to execute the workpiece coordinate system selection command (G54 to G59, G54.1) during the tilted working plane indexing when bit 6 (3TW) of parameter No. 1205 is 0, alarm PS5462, “ILLEGAL COMMAND (G68.2/G69)” is issued.

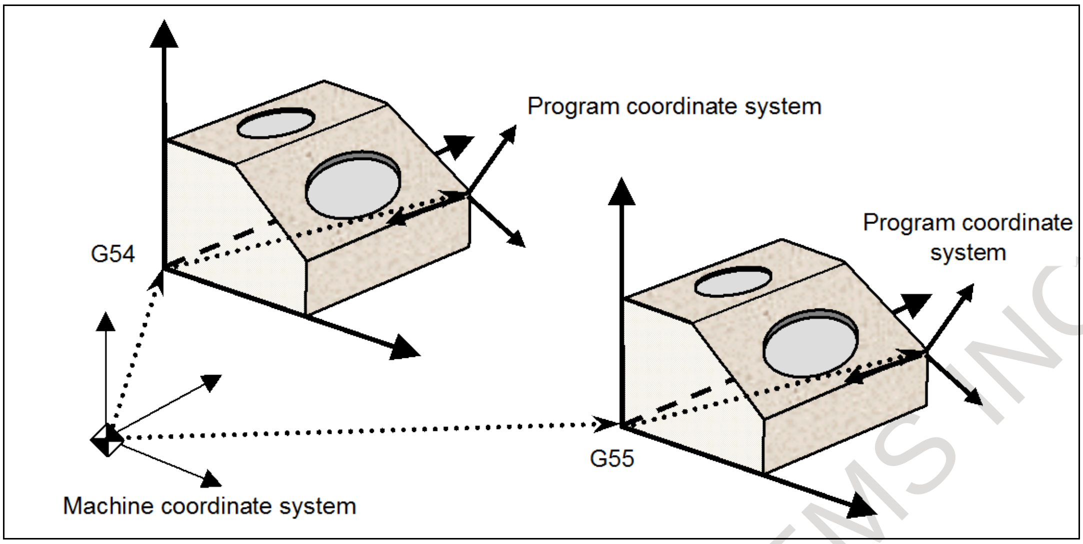


Fig. 12.16.1.2 (a) Workpiece coordinate system selection command during the tilted working plane indexing

- Minimum command unit of rotation angles

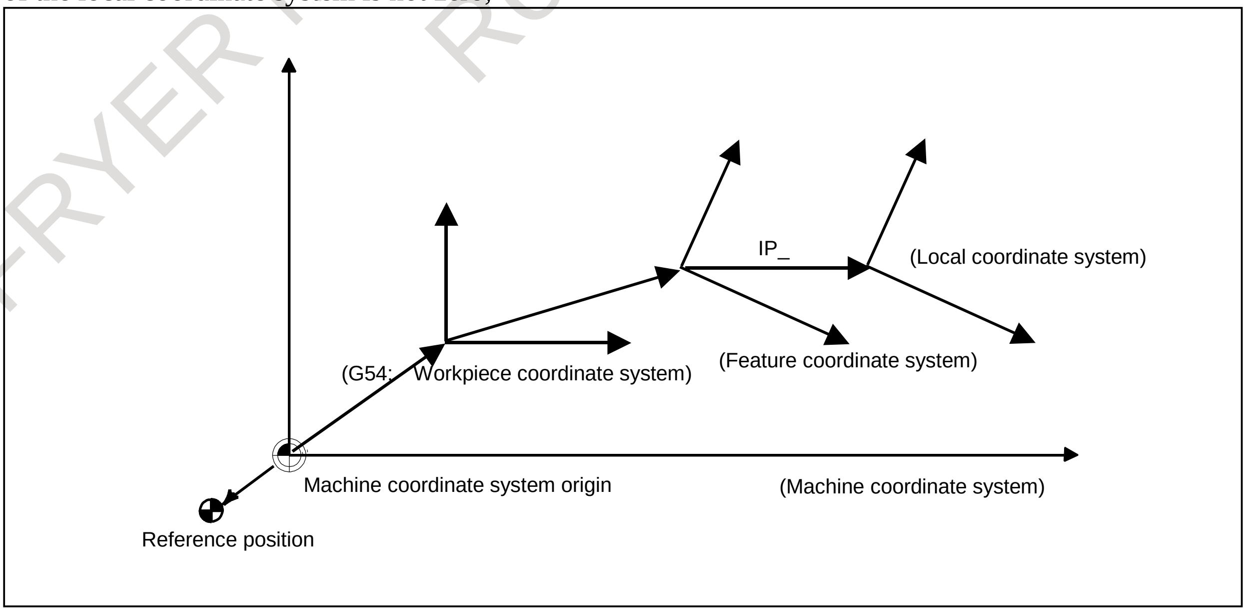
The minimum command unit of the rotation angles (I, J, K, and R) of the tilted working plane indexing is 0.001 degree regardless of the increment system. By setting bit 2 (TFR) of parameter No. 11630 to 1, the minimum command unit of the rotation angles can be set to 0.00001 degree.

- Cutting feedrate clamp

The cutting feedrate is clamped so that the feedrate of each real axis after the conversion by the tilted working plane indexing does not exceed the maximum cutting feedrate (Parameter No.1432 if look-ahead acceleration/deceleration before interpolation is enabled and parameter No.1430 otherwise).

- Local Coordinate System

The local coordinate system is available to the feature coordinate system during the tilted working plane indexing.
X,Y,Z commands of the local coordinate system command block defines a local coordinate system that the feature coordinate system is translated in X,Y,Z direction.
Alarm PS5462 occurs when the tilted working plane indexing is specified on the condition that the offset of the local coordinate system is not zero,



12.16.1.3 Tilted working plane indexing based on roll-pitch-yaw

Overview

With the tilted working plane indexing, coordinate system conversion by rotation about the X-axis, Y-axis, and Z-axis of a workpiece coordinate system in this order can be used (roll-pitch-yaw).
The order of rotary axes can be specified using address Q.

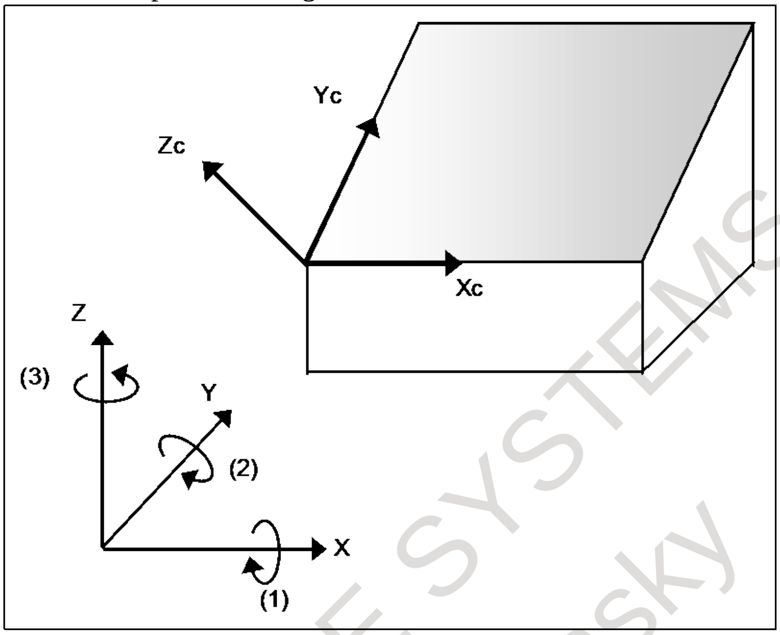


Fig. 12.16.1.3 (a) Tilted working plane indexing based on roll-pitch-yaw

Explanation

Suppose that the coordinate system is rotated about (1) the X-axis, (2) the Y-axis, and (3) the Z-axis in this order.

A "workpiece coordinate system" rotated by angle α about the X-axis is "coordinate system 1".

"Coordinate system 1" rotated by angle β about the Y-axis is "coordinate system 2". "Coordinate system 2" rotated by angle γ about the Z-axis then shifted by (X_0,Y_0,Z_0) from the workpiece coordinate system origin is a "feature coordinate system".

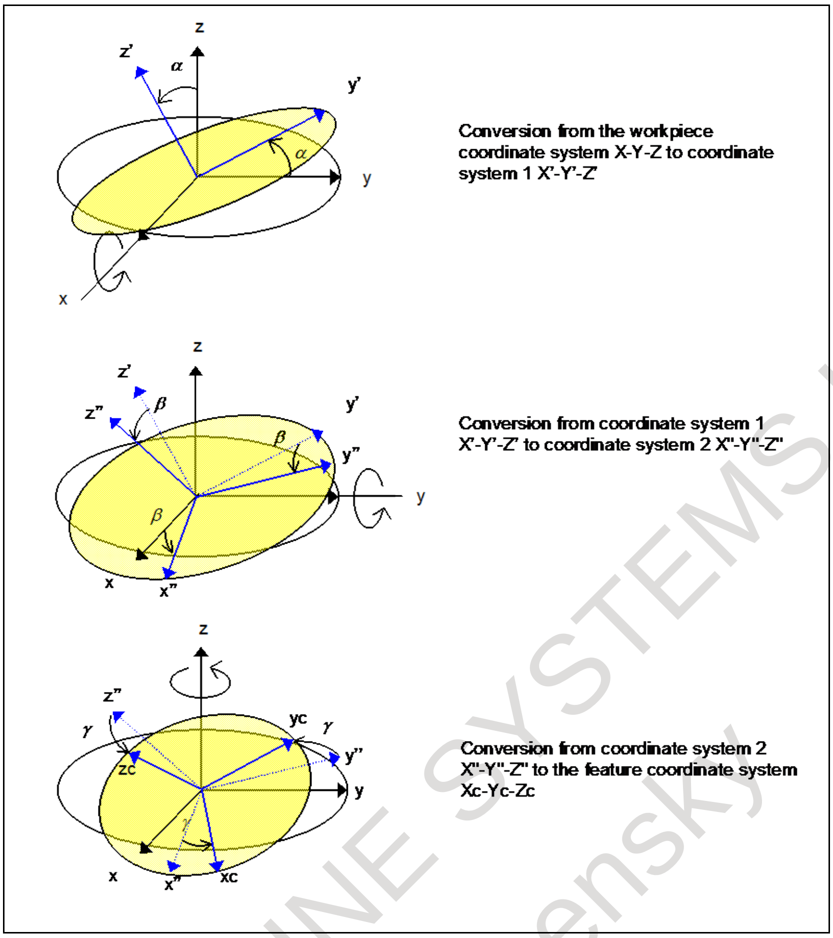


Fig. 12.16.1.3 (b) Determination of a feature coordinate system

12.16.1.4 Tilted working plane indexing based on three points

Overview

With the tilted working plane indexing, a tilted working plane can be specified by specifying three points in a feature coordinate system.

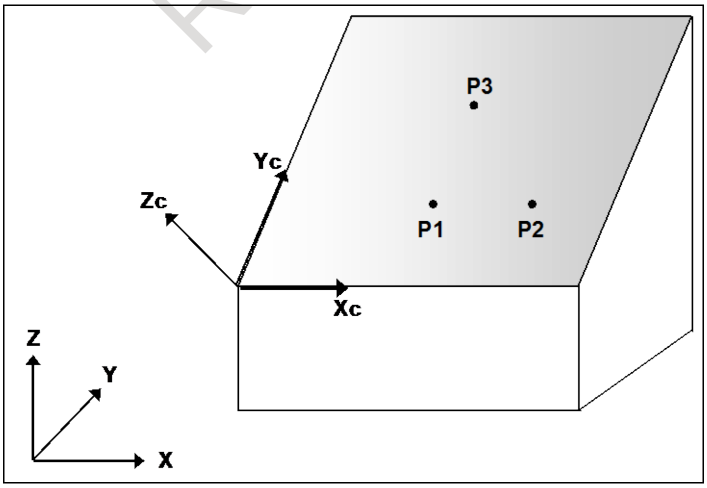


Fig. 12.16.1.4 (a) Tilted working plane indexing based on three points

Explanation

- Determination of a feature coordinate system

Three entered points are named P1, P2, and P3 in the order of entry.
The P1-to-P2 direction is defined as the X-axis of a feature coordinate system. Among the directions that are on the plane containing the three points and are normal to the X-axis of the feature coordinate system, the direction that makes a smaller angle with the P1-to-P3 vector is defined as the Y-axis of the feature coordinate system. The Z-axis of the feature coordinate system is defined according to the right-handed system.

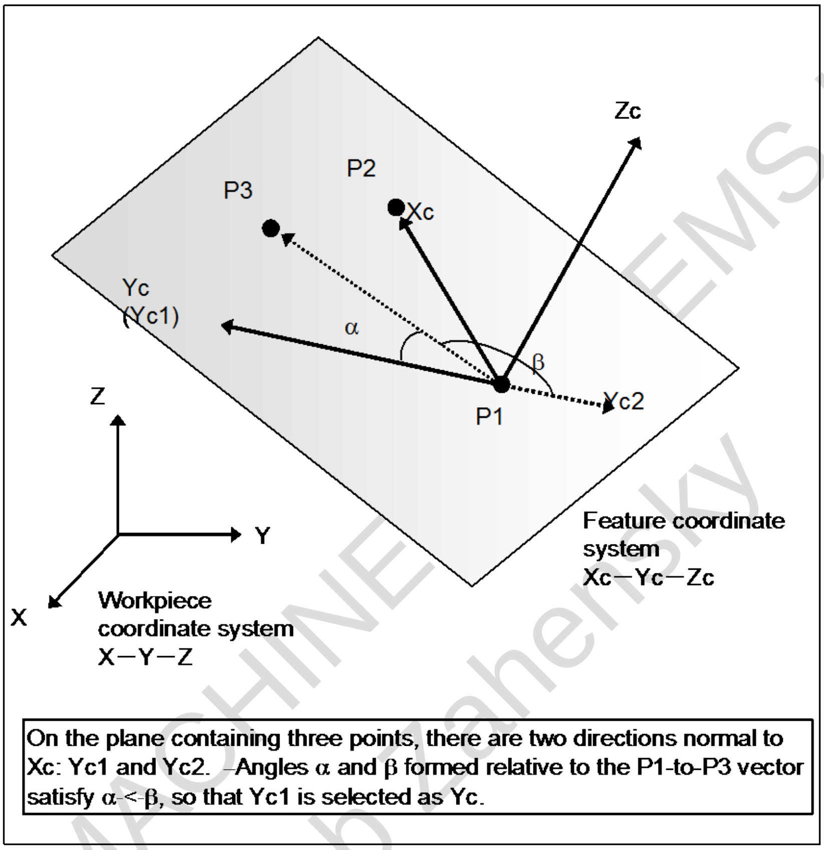


Fig. 12.16.1.4 (b) Determination of a feature coordinate system

- Origin of the feature coordinate system

The origin of the feature coordinate system is the first specified point P1.
By setting an origin shift amount (G68.2 P2 Q0 X_Y_Z_), the origin of the feature coordinate system is shifted by (X,Y,Z) from P1. Specify (X,Y,Z) in the feature coordinate system.

- Angular displacement R

The angular displacement R is positive for clockwise rotation viewed in the Z-axis direction in the feature coordinate system.

12.16.1.5 Tilted working plane indexing based on two vectors

Overview

With the tilted working plane indexing, a tilted working plane can be specified by specifying an X-axis direction vector and a Z-axis direction vector in the feature coordinate system.

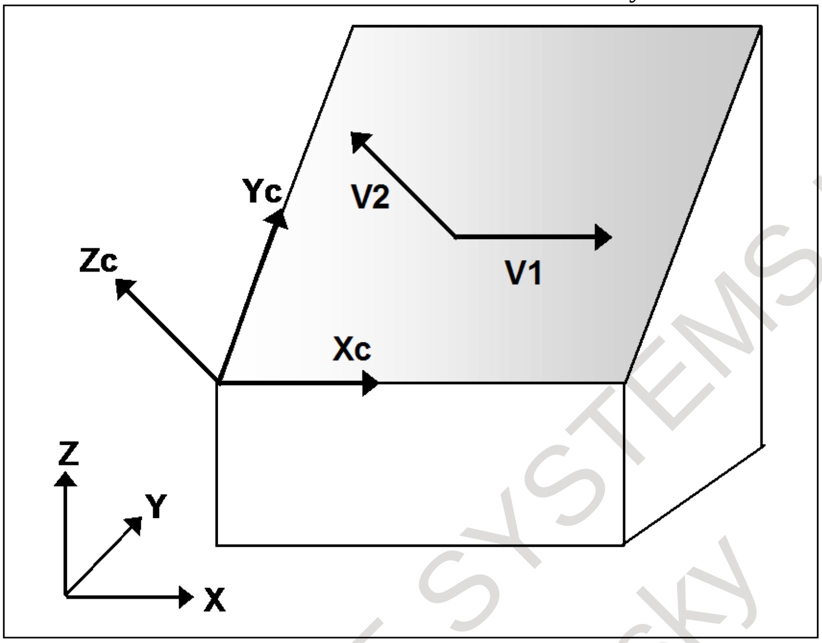


Fig. 12.16.1.5 (a) Tilted working plane indexing based on two vectors

Explanation

- Determination of a feature coordinate system

The first vector is defined as the X-axis of the feature coordinate system, and the second vector is defined as the Z-axis of the feature coordinate system. The Y-axis of the feature coordinate system is defined according to the right-handed system.

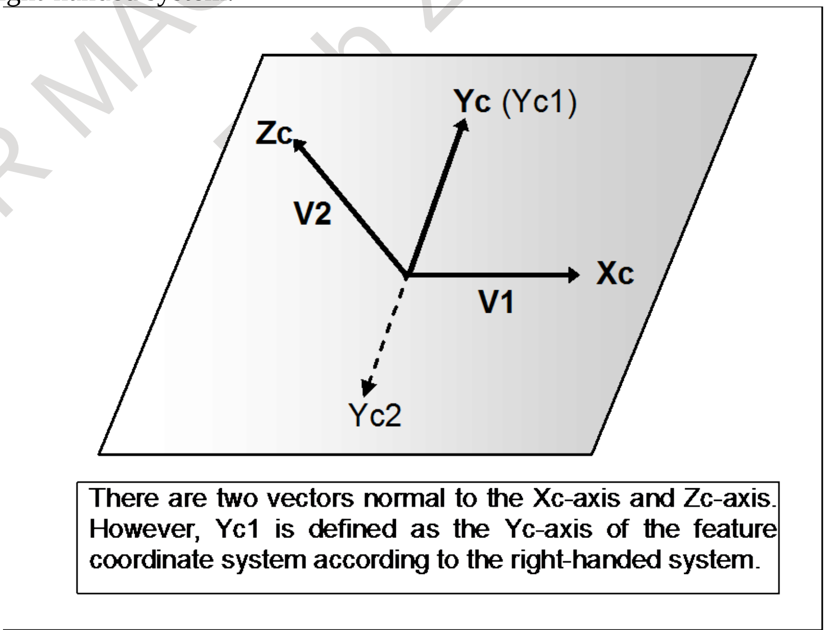


Fig. 12.16.1.5 (b) Determination of a feature coordinate system

- When the first and second vectors are not normal

When the first vector and second vector are not normal, the orthographical vector from the second vector to plane P normal to the first vector is defined as the Z-axis of the feature coordinate system.

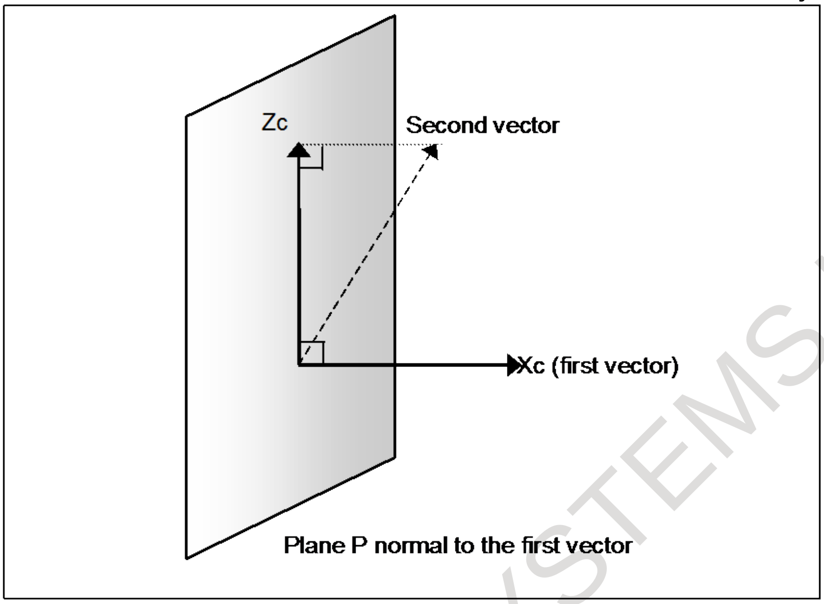


Fig. 12.16.1.5 (c) When the first and second vectors are not normal

12.16.1.6 Tilted working plane indexing based on projection angles

Overview

With the tilted working plane indexing, a tilted working plane can be specified based on projection angles.
A plane determined by vector A and vector B produced by rotating the X-axis vector and Y-axis vector of the workpiece coordinate system is defined to be a tilted working plane.

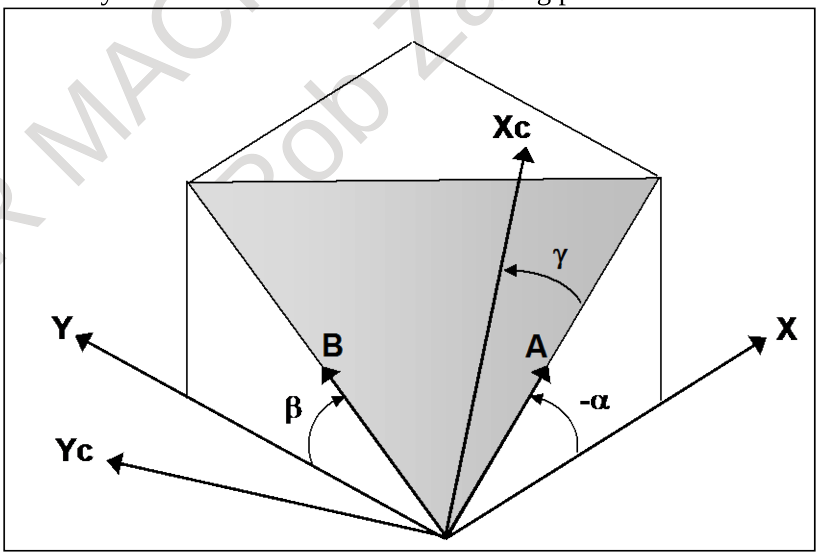


Fig. 12.16.1.6 (a) Tilted working plane indexing based on projection angles

Explanation

- Determination of a feature coordinate system

The X-axis direction vector of the workpiece coordinate system rotated by α about the Y-axis of the workpiece coordinate system is defined as vector A. The Y-axis direction vector of the workpiece coordinate system rotated by β about the X-axis of the workpiece coordinate system is defined to be vector B.

The direction normal to plane P containing vector A and vector B (direction of the outer product of A × B) is defined to be the Z-axis direction of the feature coordinate system.

Vector A rotated by γ about the Z-axis of the feature coordinate system is defined to be the X-axis direction of the feature coordinate system. The Y-axis of the feature coordinate system is defined according to the right-handed system.

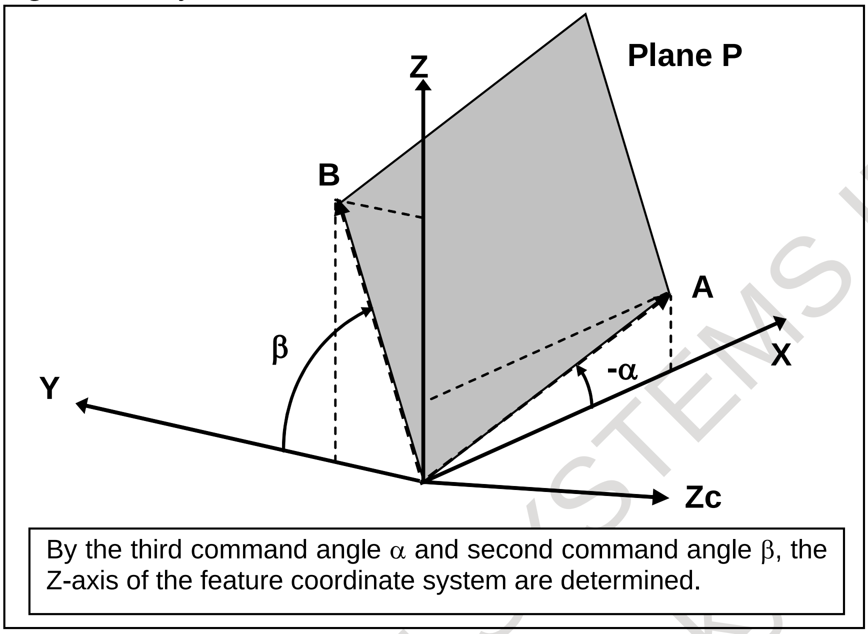


Fig. 12.16.1.6 (b) Determination of a feature coordinate system (Z axis)

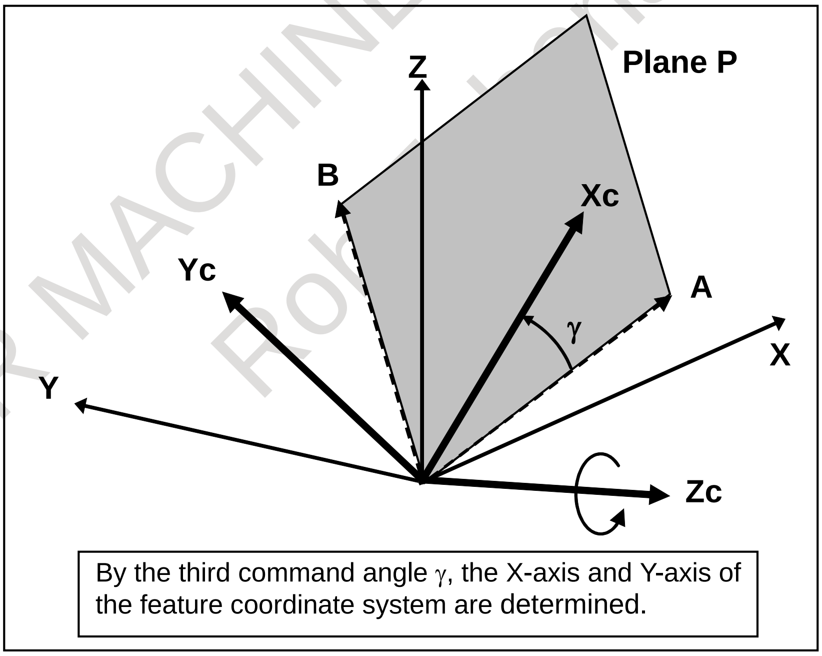


Fig. 12.16.1.6 (c) Determination of a feature coordinate system (X and Y axes)

NOTE

When vector A and vector B are considered to be parallel with each other (when the angle formed by the two vectors is smaller than 1degree), alarm PS5457, "G68.2/G68.3 FORMAT ERROR" is issued.

12.16.1.7 Tilted working plane indexing by tool axis direction

Overview

By specifying G68.3, a coordinate system (feature coordinate system) where the tool axis direction is the +Z-axis direction can be automatically specified. When a feature coordinate system is used, a program for cutting a hole or pocket in a plane tilted relative to the workpiece coordinate system can be made simpler. This function can automatically generate a feature coordinate system that is normal to the tool direction.

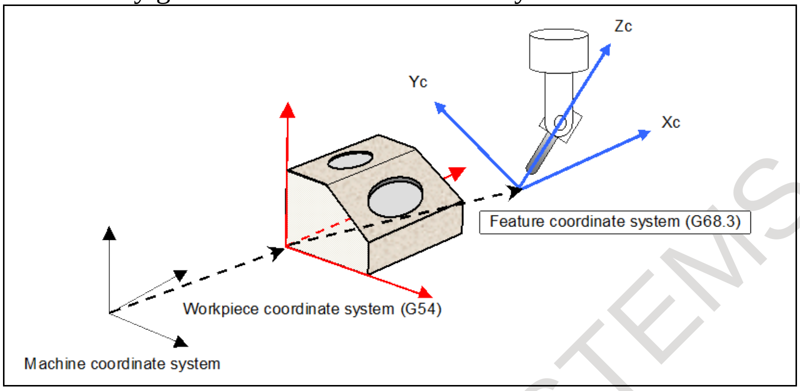


Fig. 12.16.1.7 (a) Feature coordinate system

When G68.3 is specified in a block, the coordinate system for programming is changed to a feature coordinate system. All commands after the block are regarded as commands in the feature coordinate system until G69 is specified.

Explanation

- Feature coordinate system

By specifying G68.3, a feature coordinate system with the tool axis direction being the +Z-axis direction can be created. The tool axis direction means the tool axis direction based on the rotation axis position reached by automatic operation or manual operation.

- Origin of a feature coordinate system

The origin of a feature coordinate system must be specified using an absolute command.
Even in the incremental command mode, the specified origin is regarded as an absolute position.
When 1 address or 2 addresses are omitted in X, Y, or Z, alarm PS5457, "G68.2/G68.3 FORMAT ERROR" is issued.
When the origin of a feature coordinate system is not specified, the position when G68.3 is specified is used as the origin.

- Determination of a feature coordinate system

Determination of a feature coordinate system depends on whether P1 command is present.

(1) When G68.3 block does not include P1 command (G68.3)

When G68.3 is specified, the tool axis direction vector ($\vec{T}$) represents the +Z direction ($\vec{Zc}$) of the feature coordinate system.

The vector normal to a plane formed by the +Z direction ($\vec{Zc}$) of the feature coordinate system and the vertical axis direction vector ($\vec{P}$) (parameter No. 12321) represents the +X direction ($\vec{Xc}$) of the feature coordinate system.

Expression: $\vec{Xc} = \vec{P} \times \vec{Zc}$

The vector normal to the +Z direction ($\vec{Zc}$) of the feature coordinate system and the +X direction ($\vec{Xc}$) of the feature coordinate system represents the +Y direction ($\vec{Yc}$) of the feature coordinate system.

Expression: $\vec{Yc} = \vec{Zc} \times \vec{Xc}$

When R is commanded, a coordinate system rotated by R around Zc axis from the above-mentioned coordinate system is the feature coordinate system.

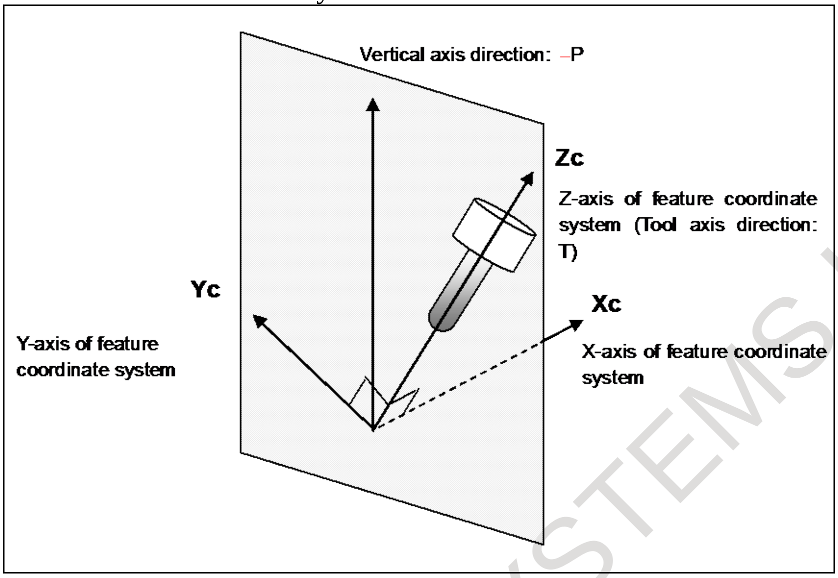


Fig. 12.16.1.7 (b) Determination of a feature coordinate system

When the tool axis direction vector ($\vec{T}$) is parallel with the vertical axis direction vector ($\vec{P}$) (parameter No. 12321) (when the angle between the vector ($\vec{T}$) and the vector ($\vec{P}$) is equal to or less than the value of parameter No. 12322), the feature coordinate system Xc-Yc-Zc is as indicated below. By specifying angular displacement R, a feature coordinate system rotated about the Z-axis of this coordinate system can be specified.

Table 12.16.1.7 (a) Feature coordinate system

Parameter No. 12321	Z-axis of feature coordinate system Zc	X-axis of feature coordinate system Xc	Y-axis of feature coordinate system Yc
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

When 0 is set in parameter No. 12321, the vertical axis direction is the reference tool axis direction (parameter No. 19697).

If a value other than 0 through 3 is set in parameter No. 12321, the alarm PS5459, "MACHINE PARAMETER INCORRECT" is issued.

NOTE

Tool axis direction is Z-axis direction of feature coordinate system regardless of the reference tool axis direction (parameter No. 19697).

- Angular displacement R

Angular displacement R is positive when a rotation is made clockwise viewed in the Z-axis direction of the feature coordinate system. The range of angular displacement R is: 0.0degree ≤ R ≤ 360.0degree.

(2) When G68.3 block includes P1 command (G68.3 P1)

G68.3 P1 command defines the feature coordinate system corresponding to tool rotation axis position. The direction of the feature coordinate system is the direction of reference coordinate system rotated by tool rotation axes.

The reference coordinate system of feature coordinate system (the feature coordinate system that is defined when absolute coordinate system of tool rotation axes is 0) is as follows by the parameter No.19697 for reference tool axis direction.

The feature coordinate system defined by "G68.3 P1" command is the coordinate system that the reference coordinate system is rotated by tool rotation axis position and reference angle RA (parameter No.19698), RB (parameter No.19699).

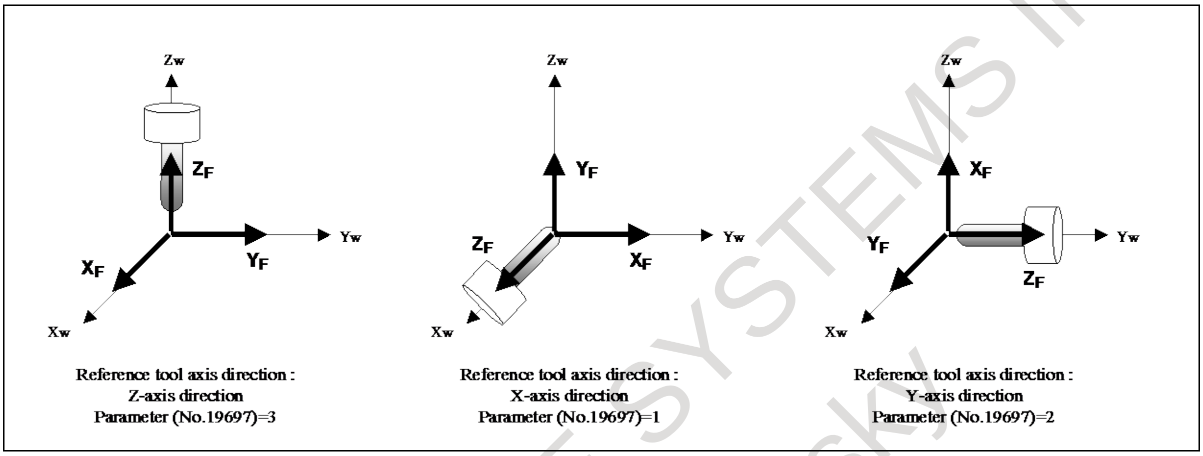
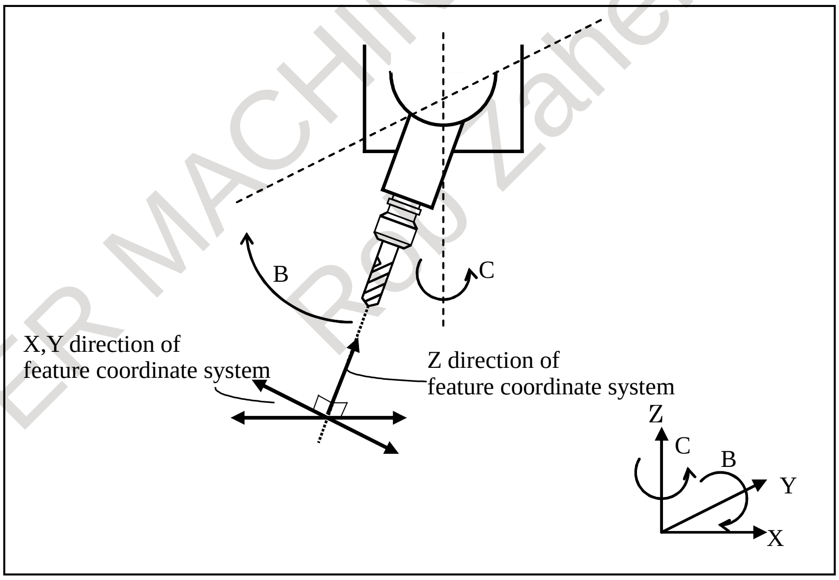


Fig. 12.16.1.7 (c) The reference coordinate system of feature coordinate system (G68.3 P1)



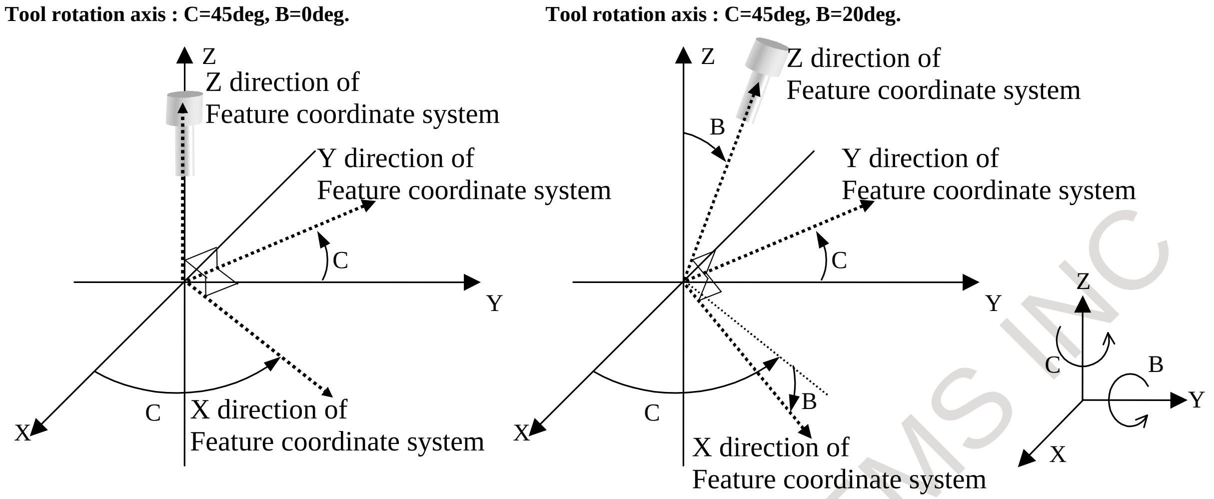


Fig. 12.16.1.7 (d) Example that reference tool axis direction is Z direction

- Machine of table rotation type

On a machine of table rotation type, the tool direction remains unchanged. So, a feature coordinate system based on the reference tool axis direction (parameter No.19697) is set. However, the origin specification of the feature coordinate system and angular displacement R about the Z-axis are valid.

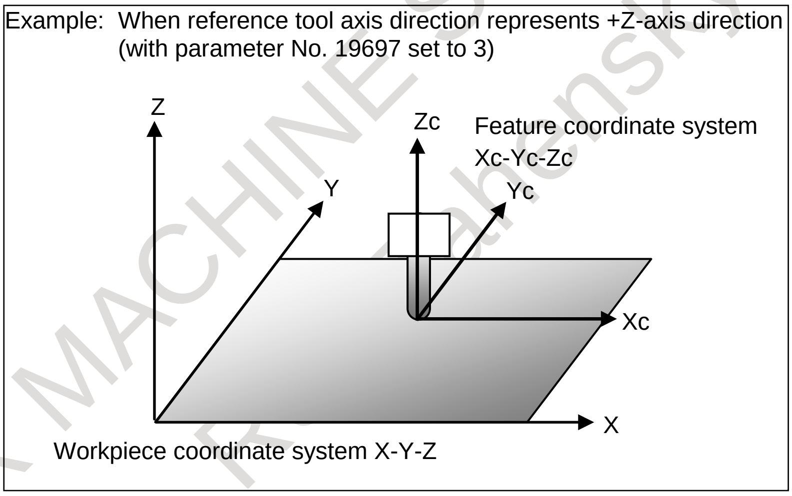


Fig. 12.16.1.7 (e) Machine of table rotation type

- G53.1 / G53.6 command in G68.3 mode

When G53.1 or G53.6 command is executed in G68.3 mode, alarm PS5458 “ILLEGAL USE OF G53.1/G53.6” occurs.

- Use in combination with tool length compensation

The G68.3 command can be specified even during tool length compensation.

- Example of operation

An example of operation on a machine of tool rotation type is given below.
The machine configuration is "BC type reference tool axis Z-axis".

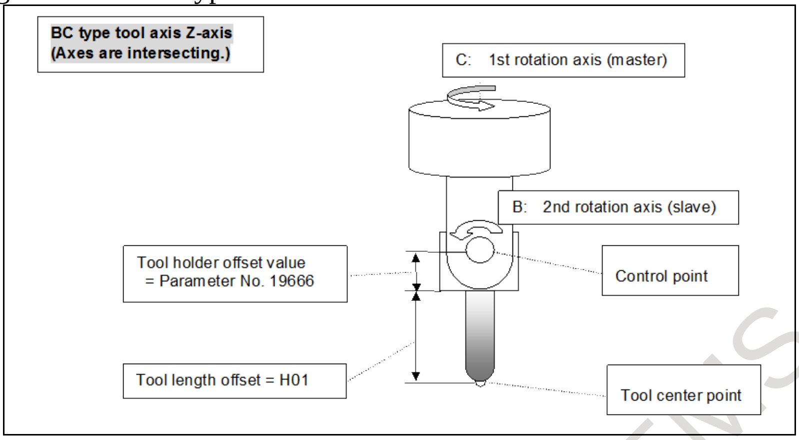


Fig. 12.16.1.7 (f) Example of operation on a machine of tool rotation type

Sample program 1

```
O0100 ;
N1 G55 ;
N2 G90 G01 X0Y0Z50.0 F1000 ;
N3 G43 H01 X0 Y0 Z0 ;
N4 B-45.0 ;
N5 G68.3 ;
:
N6 G69 ;
:
```

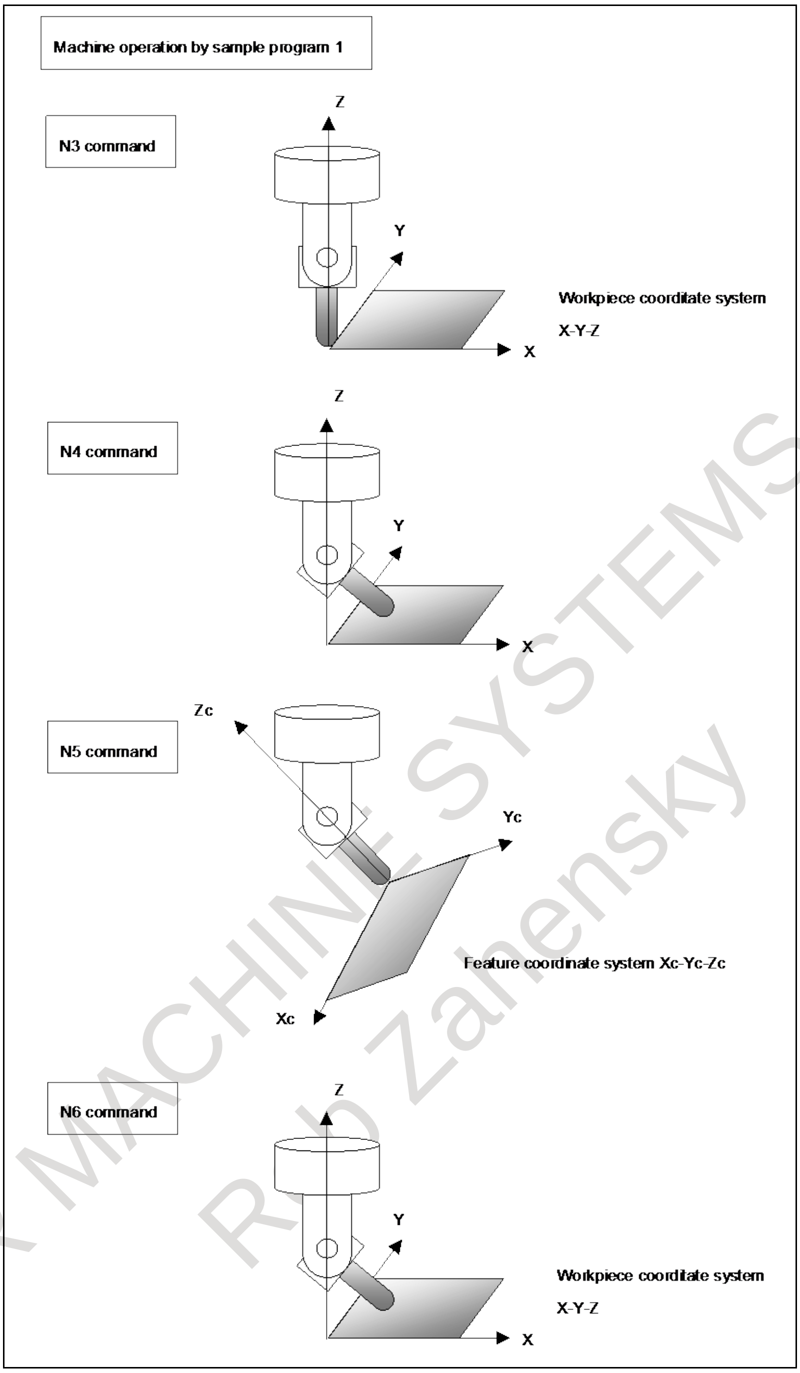


Fig. 12.16.1.7 (g) Machine operation with sample program 1

N3 block: Performs tool length compensation in the workpiece coordinate system.
The tool center point moves to the origin of the workpiece coordinate system.
N4 block: Tilts the tool.
N5 block: Sets a feature coordinate system where the tool axis direction is the Z-axis direction and the tool center point is placed at the origin.
N6 block: Cancels the feature coordinate system to return to the workpiece coordinate system.

Sample program 2

```
O0100 ;
N1 G54 G90 G00 B0 C0 ;
N2 B45.0 ;
N3 C60.0 ;
N4 G68.3 P1 X0 Y0 Z0 ;
```

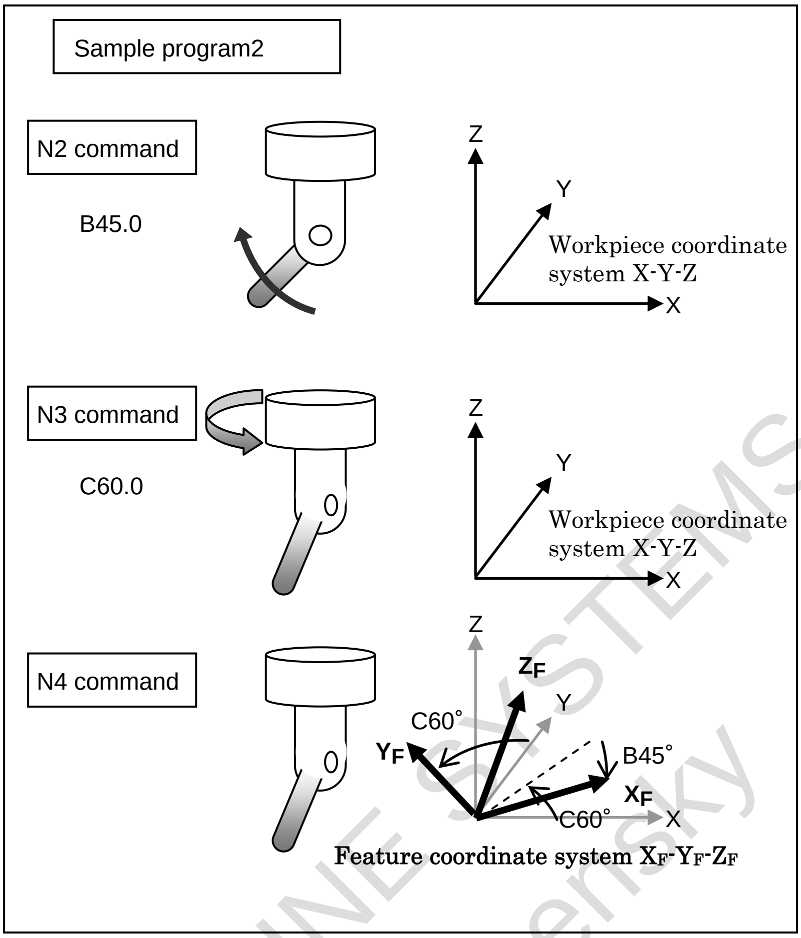


Fig. 12.16.1.7 (h)

N2 block: Tilts the tool. (B45 deg)
N3 block: Tilts the tool. (C60 deg)
N4 block: The direction of the reference coordinate system of feature coordinate system is the direction of workpiece coordinate system because the reference tool axis direction is Z direction. The feature coordinate system is the coordinate system that the reference coordinate system is rotated by 45 deg. around Y direction and 60 deg. around Z direction.

XF : The direction that X direction of workpiece coordinate system is rotated by 60 deg. around Z direction after by 45 deg. around Y direction of workpiece coordinate system.
YF : The direction that Y direction of workpiece coordinate system is rotated by 60 deg. round Z direction of workpiece coordinate system.
ZF : The direction that Z direction of workpiece coordinate system is rotated by 60 deg. around Z direction after by 45 deg. around Y direction of workpiece coordinate system.

- Multiple G68.3

After the tool axis direction is changed in G68.3 mode, by specifying G68.3, a new feature coordinate system where the tool axis direction is the +Z-axis direction can be specified.

Example of operation

An example of operation on a machine of tool rotation type is given below.
The machine configuration is "BC type reference tool axis Z-axis".

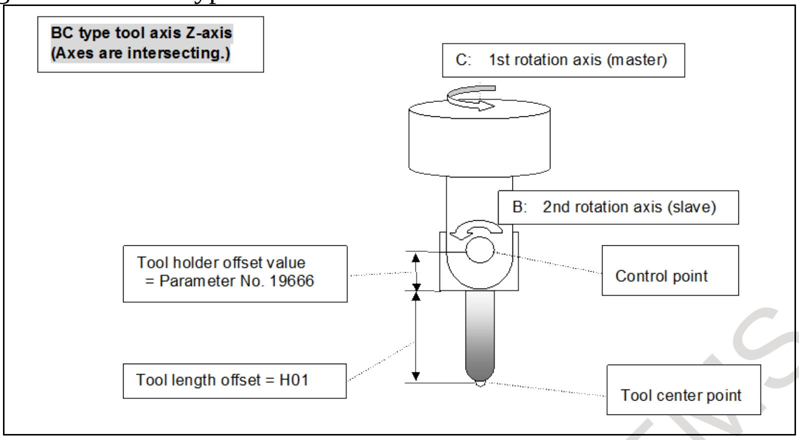


Fig. 12.16.1.7 (i) Example of operation on a machine of tool rotation type

Sample program 2

```
O0200 ;
N1 G55 ;
N2 G01 A90.0 F1000 ;
N3 G68.3 X0 Y0 Z0 R0;
:
N4 X10.0 Y0 Z0 ;
N5 C90.0;
N6 G68.3 X10.0 Y0 Z0 ;
:
N7 G69 ;
:
```

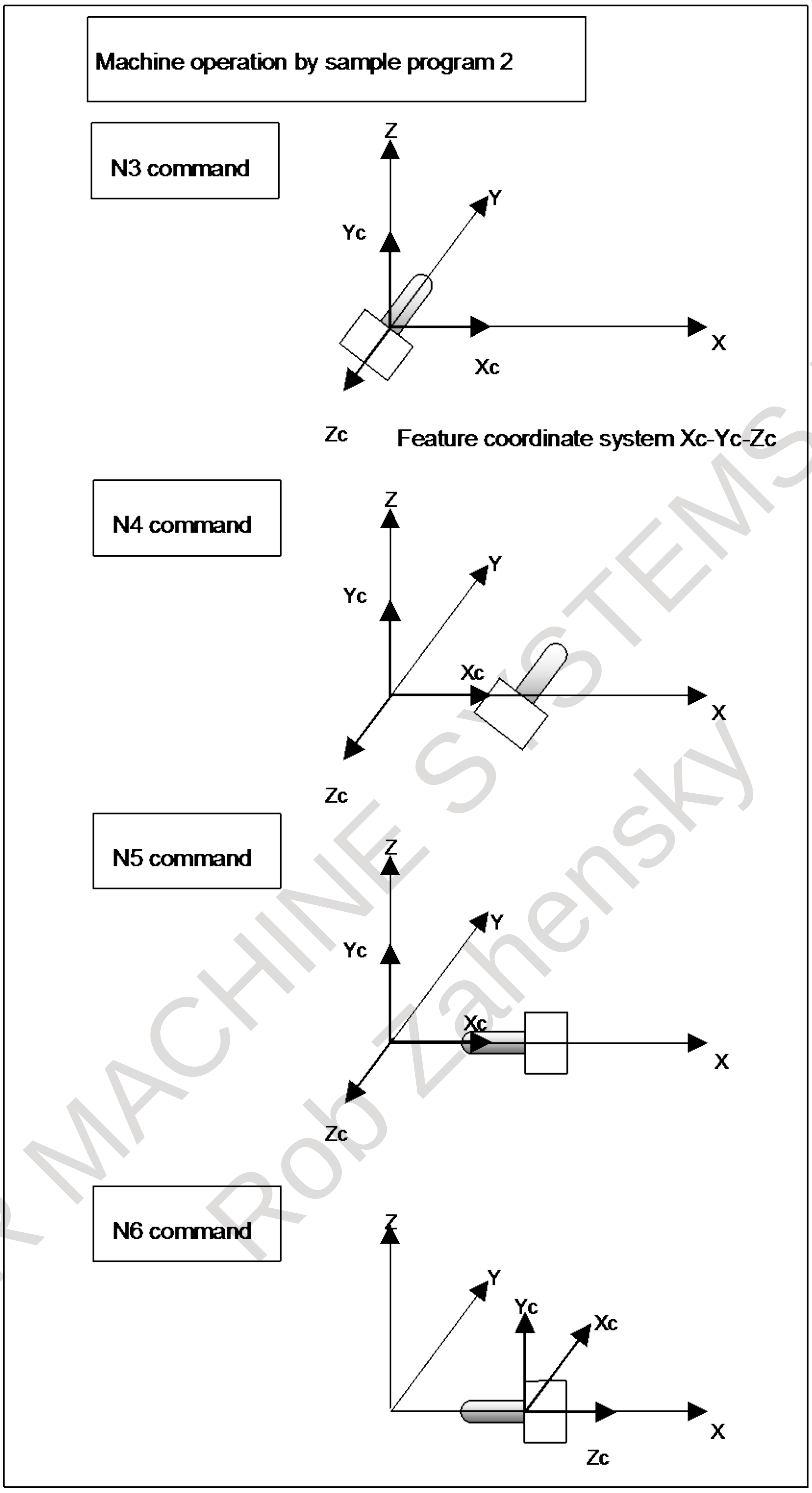


Fig. 12.16.1.7 (j) Machine operation with sample program 2

N3 block: Sets a feature coordinate system according to the tool direction.
N4 block: Specifies coordinates in the feature coordinate system.
N5 block: Changes the tool direction.
N6 block: Sets a feature coordinate system according to the tool direction.

12.16.2 Multiple Command of Tilted Working Plane Indexing

12.16.2.1 Absolute multiple command

By additionally specifying G68.2 in the tilted working plane indexing mode, a feature coordinate system produced by additionally applying coordinate system conversion to the workpiece coordinate system can be set. The workpiece coordinate system is resumed by specifying G69.
This function is enabled by setting bit 0 (MTW) of parameter No. 11221.

Format

The format of the tilted working plane indexing (G68.2) is applicable.
Specify the origin of a feature coordinate system in the workpiece coordinate system.

Example of operation

An example of operation on a tool rotation type machine is explained below.
The machine configuration is "BC type with the reference tool axis being the Z-axis".

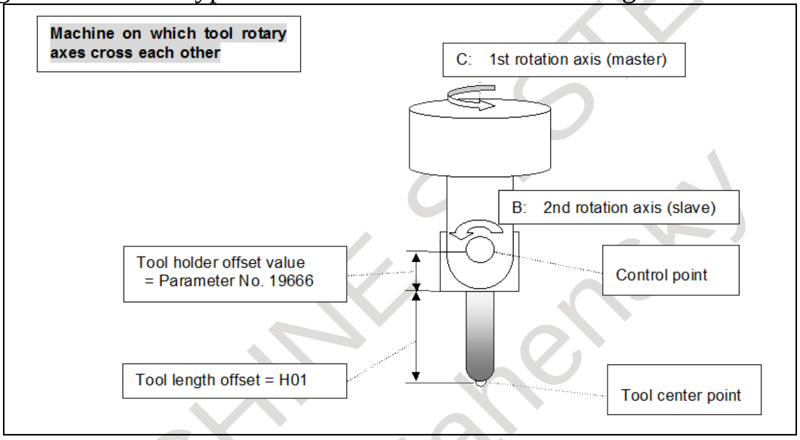


Fig. 12.16.2.1 (a) Example of operation on a machine of tool rotation type

Sample program 1

```
O0100 ;
N1 G55 ;
N2 G90 G01 X20.0 Y5.0 Z0 F1000 ;
N3 G68.2 X20.0 Y5.0 Z0 I0 J90.0 K0 ;
N4 G53.1 ;
  :
N5 X-15.0 Y0 Z-15.0 ;
N6 G68.2 X5.0 Y20.0 Z0 I90.0 J90.0 K0 ;
N7 G53.1;
  :
N8 G69 ;
  :
```

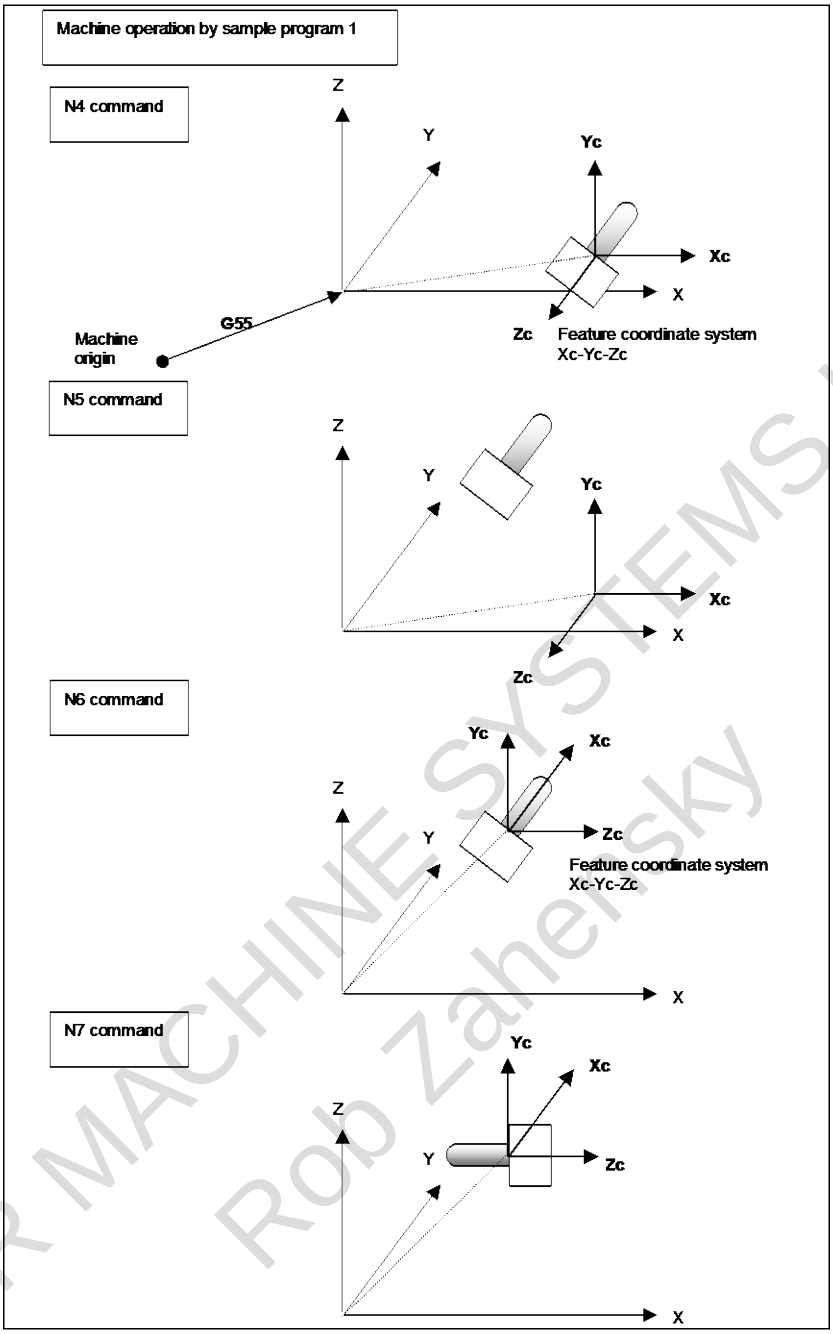


Fig. 12.16.2.1 (b) Machine operation with sample program 1

N4 block: Rotates the tool in the Z-axis direction in the feature coordinate system.
N5 block: Specifies coordinates in the feature coordinate system.
N6 block: Sets a new feature coordinate system.
N7 block: Rotates the tool in the Z-axis direction in the new feature coordinate system.

12.16.2.2 Incremental multiple command

By specifying G68.4, coordinate system conversion can be applied to the currently set feature coordinate system.
This function is enabled by setting bit 0 (MTW) of parameter No. 11221.

Format

The format of the tilted working plane indexing (G68.2) is applicable.
Specify the origin of a feature coordinate system in the immediately preceding feature coordinate system.

Table 12.16.2.2 (a) Incremental multiple command format

Specification method	Incremental multiple command
Eulerian angle	G68.4
Roll-pitch-yaw	G68.4 P1
Three points	G68.4 P2
Two vectors	G68.4 P3
Projection angles	G68.4 P4

Example of operation

An example of operation on a tool rotation type machine is explained below.
Rotary axis C rotates about the Z-axis (master axis).
Rotary axis B rotates about the Y-axis (slave axis).

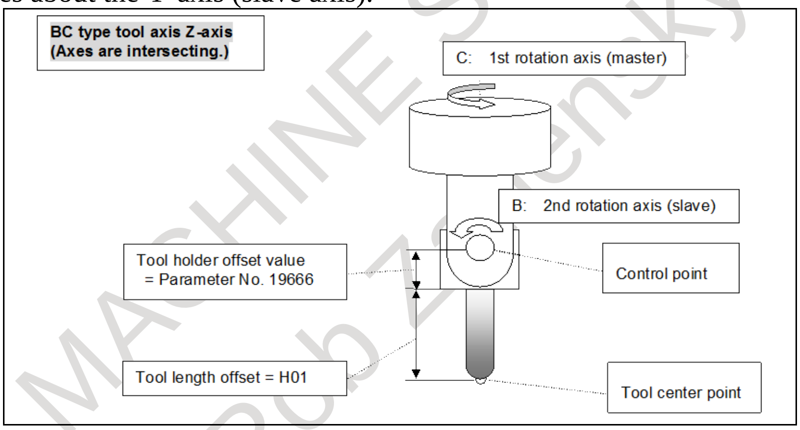


Fig. 12.16.2.2 (a) Example of operation on a machine of tool rotation type

Sample program 2
O0200 ;
N1 G55 ;
N2 G90 G01 X20.0 Y5.0 Z0 F1000 ;
N3 G68.2 X20.0 Y5.0 Z0 I0 J90.0 K0 ;
N4 G53.1 ;
:
N5 X-15.0 Y0 Z-15.0 ;
N6 G68.4 X-15.0 Y0 Z-15.0 I90.0 J90.0 K-90.0 ;
N7 G53.1;
:
N8 G69 ;
:

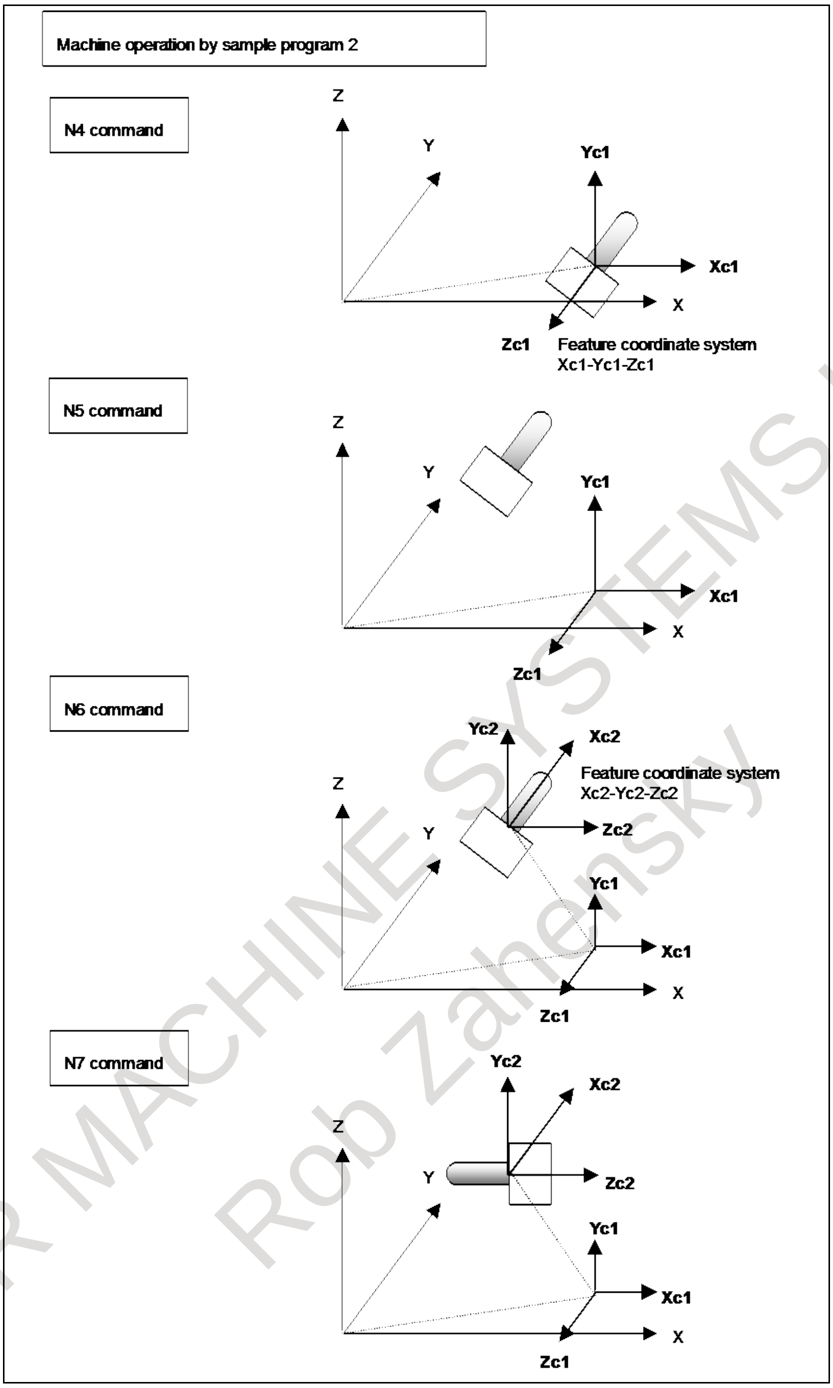


Fig. 12.16.2.2 (b) Machine operation with sample program 2

N4 block: Rotates the tool in the Z-axis direction in the feature coordinate system.
N5 block: Specifies coordinates in the feature coordinate system.
N6 block: Applies coordinate system conversion to the feature coordinate system based on Eulerian angle to set a new feature coordinate system.
N7 block: Rotates the tool in the Z-axis direction in the new feature coordinate system.

12.16.3 Tool Axis Direction Control

12.16.3.1 Tool axis direction control

G53.1 automatically specifies the +Z direction of the feature coordinate system as the tool axis direction.

Example of operation

The following gives an operation example in the machine configuration below.

- Table rotation type
- Master axis: About the Y-axis (B axis)
- Slave axis: About the Z-axis (C axis)
- Reference tool axis direction: Z direction
- Increment system for the rotation axis: 1/10(IS-C)

Program example 1

G68.2 I90.0 J0.0005 K-90.0 (rotation by 0.0005 degree about the Y-axis)
G53.1
The rotation axis position after the G53.1 command is as follows:

- Bit 2 (TFR) of parameter No. 11630 is set to 0 (minimum command unit of the rotation angles: 0.001 degree):
  B axis: 0.0010 degree
  C axis: 0.0000 degree
- Bit 2 (TFR) of parameter No. 11630 is set to 1 (minimum command unit of the rotation angles: 0.00001 degree):
  B axis: 0.0005 degree
  C axis: 0.0000 degree

Program example 2

G68.2 I90.0 J10 K-90.0 (rotation by (minimum command unit of the rotation angles × 10) about the Y-axis)
G53.1
The rotation axis position after the G53.1 command is as follows:

- Bit 2 (TFR) of parameter No. 11630 is set to 0 (minimum command unit of the rotation angles: 0.001 degree):
  B axis: 0.0100 degree
  C axis: 0.0000 degree
- Bit 2 (TFR) of parameter No. 11630 is set to 1 (minimum command unit of the rotation angles: 0.00001 degree):
  B axis: 0.0001 degree
  C axis: 0.0000 degree

About parameters

(1) Machine configuration

When parameters are set, it is important to determine the target machine configuration for parameter setting.
The following explains the machine configuration.

- Master and slave

When there are two rotation axes for controlling the orientation of a tool or two rotation axes for controlling the orientation of a table, a typical structure is such that a rotation mechanism is on the tip of another rotation mechanism. The former rotation mechanism is called the slave, and the latter is called the master.
The master and slave are sometimes called the first axis and second axis, respectively.

The rotation center axis for the master is called the first rotation axis, while the rotation center axis for the slave is called the second rotation axis. (Fig. 12.16.3.1 (a))

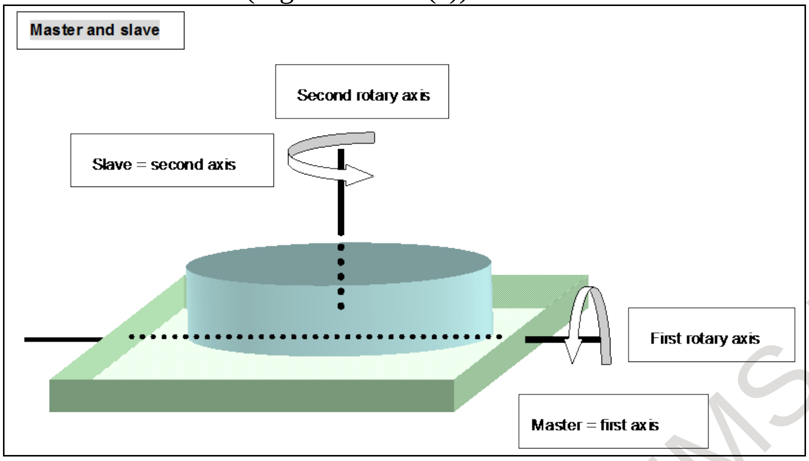


Fig. 12.16.3.1 (a) Relationships between the master and slave

- When the rotation axes of the table do not intersect

Explained below is a mechanism in which the table rotation centers do not intersect.
In the mechanism shown in the following example, the master and slave do not intersect each other. (Fig. 12.16.3.1 (b))
When both the master and slave are at 0 degrees, a vector from the origin of the machine coordinate system to a point on the first rotation axis of the table is set as a rotary table position in parameters Nos. 19700 to 19702.
Similarly, when both the master and slave are at 0 degrees, a vector from the above point to a point on the second rotation axis is set as an intersection offset vector between the first and second rotation axes of the table in parameters Nos. 19703 to 19705.

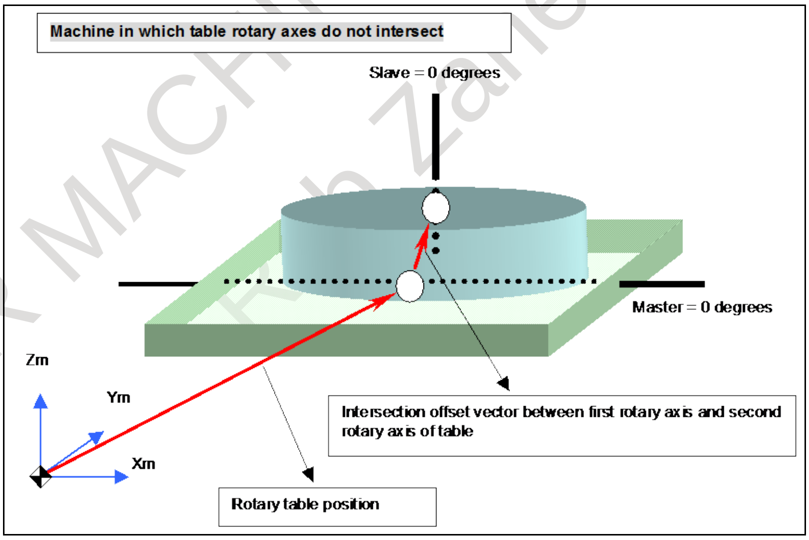


Fig. 12.16.3.1 (b) Mechanism in which the master and slave do not intersect

- When the first rotation axis of the tool and the tool axis do not intersect

Explained below is a mechanism in which the tool axis (spindle rotation center axis) and the first rotation axis of the tool do not intersect.
When both the master and slave are at 0 degrees, a vector from a point at a distance of the tool length compensation value plus the tool holder offset value from the tool center point along the tool axis to a point on the first rotation axis of the tool is set as an intersection offset vector between the tool axis and the first rotation axis of the tool in parameters Nos. 19709 to 19711.

- When the rotation axes of the tool do not intersect

Explained below is a mechanism in which the rotation axes of the tool do not intersect each other.

The Fig. 12.16.3.1 (c) is an example of a mechanism in which the first and second rotation axes of the tool do not intersect.

When both the master and slave are at 0 degrees, a vector from a point on the second rotation axis of the tool to a point on the first rotation axis of the tool is set as an intersection offset vector between the second rotation axis of the tool and the first rotation axis of the tool in parameters Nos. 19712 to 19714.

The controlled point can be shifted by setting a shift vector from the point mentioned above in parameters.

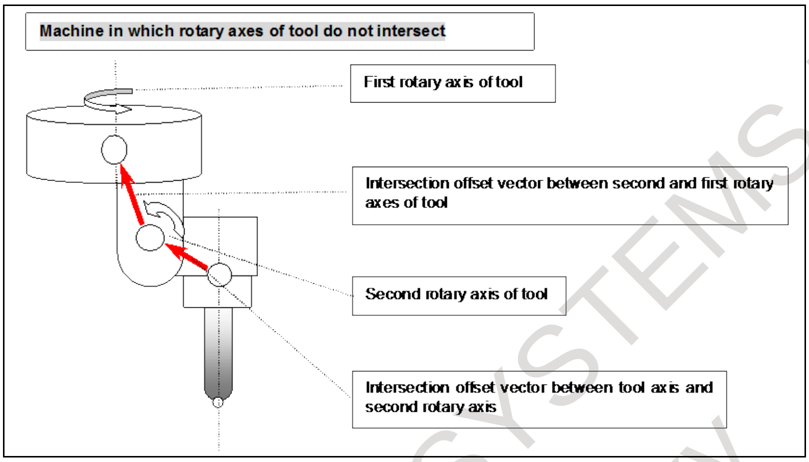


Fig. 12.16.3.1 (c) Mechanism in which the rotation axes of the tool do not intersect

(2) Examples of setting parameters

There are many parameters related to this function.

Therefore, it is recommended that parameters for a target machine be set based on some examples. These examples are given below.

<1> Example of setting parameters for a composite type machine

In the machine explained in this example, the first axis is X, the second axis is Y, the third axis is Z, the fourth axis is A, the fifth axis is B, and the sixth axis is C.

Fig. 12.16.3.1 (d) is an example of setting parameters for a composite type machine.

Rotation axis A is a table rotation axis on the X-axis.

Rotation axis B is a tool rotation axis on the Y-axis.

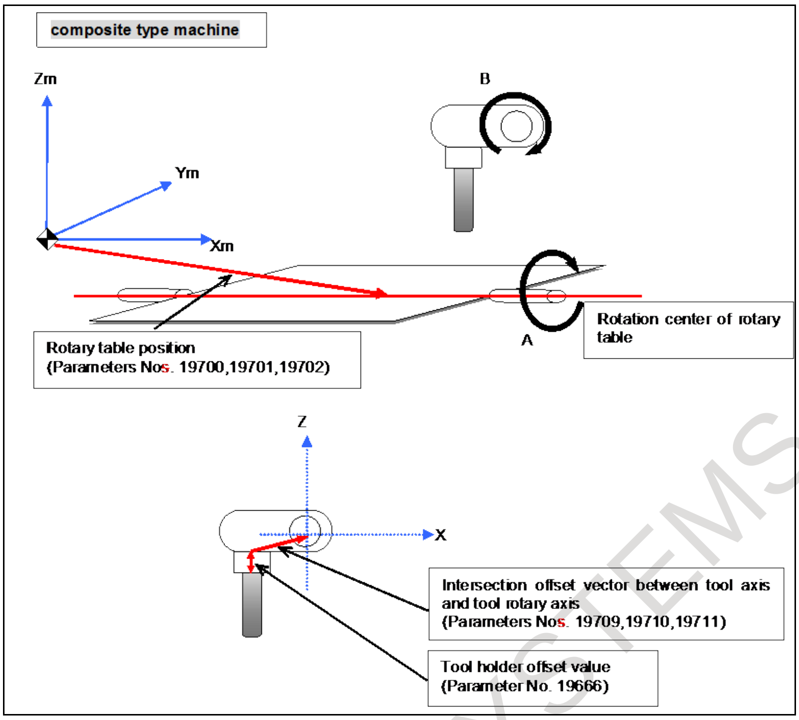


Fig. 12.16.3.1 (d) Composite type machine example

In the above composite type machine (Fig. 12.16.3.1 (d)), if rotation axis B is at 0 degrees, assume the following:
Tool holder offset value = 2.0
Rotary table position = (5.0, 2.0, -15.0)
Intersection offset vector between the tool axis and tool rotation axis = (3.0, 0, 2.0)

Then, the parameters for the above composite type machine are set as follows:

Table 12.16.3.1 (a) Example of setting parameters for a composite type machine

Parameter No.	Setting example	Description
19665#4	0	Automatic calculation for controlled-point shifting
19665#5	0	Controlled-point shift
19666	2.0	Tool holder offset value
19667	X0.0 Y0.0 Z0.0	Controlled-point shift vector
19680	21	Mechanical unit type
19681	5(B)	Controlled axis number for the first rotation axis
19682	2(Y)	Axis direction of the first rotation axis
19684	0	Rotation direction of the first rotation axis
19685	0.0	Rotation angle when the first rotation axis is a hypothetical axis
19686	4(A)	Controlled axis number for the second rotation axis
19687	1(X)	Axis direction of the second rotation axis
19689	1	Rotation direction of the second rotation axis
19690	0.0	Rotation angle when the second rotation axis is a hypothetical axis
19696#0	0	The first rotation axis is an ordinary rotation axis.
19696#1	0	The second rotation axis is an ordinary rotation axis.
19697	3	Direction of the reference tool axis
19700	5.0	Rotary table position (X-axis)
19701	2.0	Rotary table position (Y-axis)
19702	-15.0	Rotary table position (Z-axis)

Parameter No.	Setting example	Description
19703	0.0	Intersection offset vector between the first and second rotation axes of the table (X-axis)
19704	0.0	Intersection offset vector between the first and second rotation axes of the table (Y-axis)
19705	0.0	Intersection offset vector between the first and second rotation axes of the table (Z-axis)
19709	3.0	Intersection offset vector between the tool axis and the first rotation axis of the tool (X-axis)
19710	0.0	Intersection offset vector between the tool axis and the first rotation axis of the tool (Y-axis)
19711	2.0	Intersection offset vector between the tool axis and the first rotation axis of the tool (Z-axis)
19712	0.0	Intersection offset vector between the second and first rotation axes of the tool (X-axis)
19713	0.0	Intersection offset vector between the second and first rotation axes of the tool (Y-axis)
19714	0.0	Intersection offset vector between the second and first rotation axes of the tool (Z-axis)

⚠ **CAUTION (Machine configurations resulting in an alarm)**

In the machine settings shown below, there may be no angle solution when G53.1 is issued.
So, specifying G53.1 results in an alarm.
<1> The table rotation axis and tool rotation axis are on the same axis.
Example: Parameter No. 19682 = parameter No. 19687 = 1 (X)
<2> The tool axis direction and tool rotation axis are in the same axis direction.
Example: Parameter No. 19682 = parameter No. 19697 = 3 (Z)

<2> Example of setting parameters for a tool rotation type machine

Fig. 12.16.3.1 (e) is an example of setting parameters for a tool rotation type machine.
Rotation axis C is a tool rotation axis (master) on the Z-axis.
Rotation axis B is a tool rotation axis (slave) on the Y-axis.

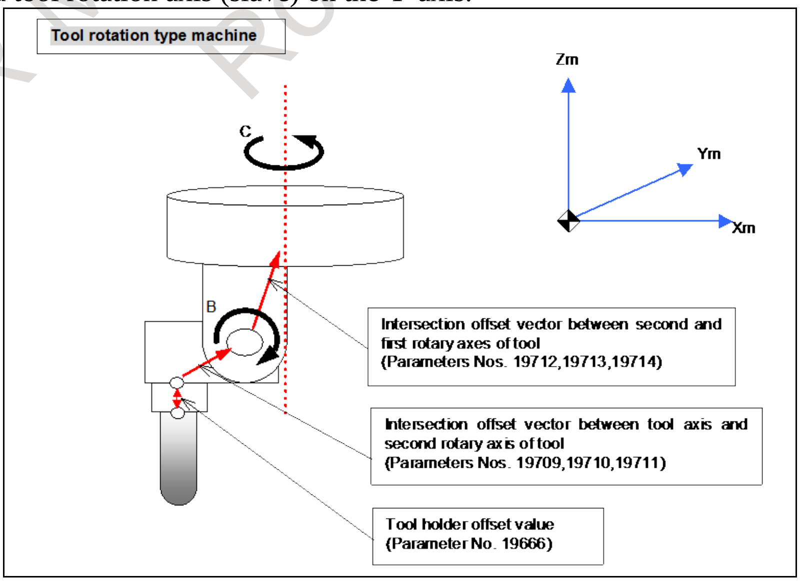


Fig. 12.16.3.1 (e) Tool rotation type machine example

In the above tool rotation type machine (Fig. 12.16.3.1 (e)), if rotation axes B and C are at 0 degrees, assume the following:
Tool holder offset value = 2.0
Intersection offset vector between the tool axis and the second rotation axis of the tool = (5.0,0,3.0)
Intersection offset vector between the second and first rotation axes of the tool = (3.0,3.0,8.0)

Then, the parameters for the above tool rotation type machine are set as follows:

Table 12.16.3.1 (b) Setting parameters for a tool rotation type machine

Parameter No.	Setting example	Description
19665#4	0	Automatic calculation for controlled-point shifting
19665#5	0	Controlled-point shift
19666	2.0	Tool holder offset value
19667	X0.0 Y0.0 Z0.0	Controlled-point shift vector
19680	2	Mechanical unit type
19681	6(C)	Controlled axis number for the first rotation axis
19682	3(Z)	Axis direction of the first rotation axis
19684	0	Rotation direction of the first rotation axis
19685	0.0	Rotation angle when the first rotation axis is a hypothetical axis
19686	5(B)	Controlled axis number for the second rotation axis
19687	2(Y)	Axis direction of the second rotation axis
19689	0	Rotation direction of the second rotation axis
19690	0.0	Rotation angle when the second rotation axis is a hypothetical axis
19696#0	0	The first rotation axis is an ordinary rotation axis.
19696#1	0	The second rotation axis is an ordinary rotation axis.
19697	3	Direction of the reference tool axis
19700	0.0	Rotary table position (X-axis)
19701	0.0	Rotary table position (Y-axis)
19702	0.0	Rotary table position (Z-axis)
19703	0.0	Intersection offset vector between the first and second rotation axes of the table (X-axis)
19704	0.0	Intersection offset vector between the first and second rotation axes of the table (Y-axis)
19705	0.0	Intersection offset vector between the first and second rotation axes of the table (Z-axis)
19709	5.0	Intersection offset vector between the tool axis and the first rotation axis of the tool (X-axis)
19710	0.0	Intersection offset vector between the tool axis and the first rotation axis of the tool (Y-axis)
19711	3.0	Intersection offset vector between the tool axis and the first rotation axis of the tool (Z-axis)
19712	3.0	Intersection offset vector between the second and first rotation axes of the tool (X-axis)
19713	3.0	Intersection offset vector between the second and first rotation axes of the tool (Y-axis)
19714	8.0	Intersection offset vector between the second and first rotation axes of the tool (Z-axis)

> ⚠ **CAUTION (Machine configurations resulting in an alarm)**
> In the machine settings shown below, there may be no angle solution when G53.1 is issued.
> So, specifying G53.1 results in an alarm.
> <1> The first rotation axis (master) and second rotation axis (slave) of the tool are on the same axis.
> Example: Parameter No. 19682 = parameter No. 19687 = 1 (X)
> <2> The tool axis direction and the second rotation axis (slave) of the tool are in the same axis direction.
> Example: Parameter No. 19697 = parameter No. 19687 = 3 (Z)

<3> Example of setting parameters for a table rotation type machine

Fig. 12.16.3.1 (f) is an example of setting parameters for a table rotation type machine.
Rotation axis A is a table rotation axis (master) on the X-axis.
Rotation axis B is a table rotation axis (slave) on the Y-axis.

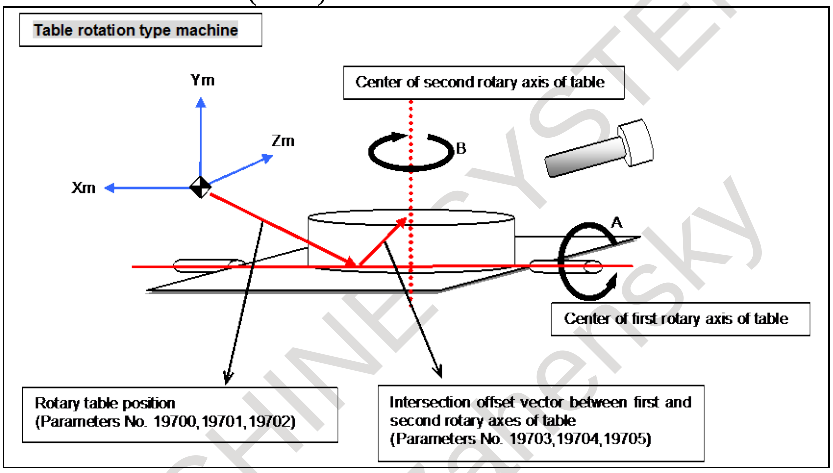


Fig. 12.16.3.1 (f) Table rotation type machine example

In the above table rotation type machine (Fig. 12.16.3.1 (f)), if rotation axes A and B are at 0 degrees, assume the following:
Tool holder offset value = 2.0
Rotary table position = (-10.0,-30.0,0)
Intersection offset vector between the first and second rotation axes of the table = (0,8.0,8.0)

Then, the parameters for the above table rotation type machine are set as follows:

Table 12.16.3.1 (c) Setting parameters for a table rotation type machine

Parameter No.	Setting example	Description
19665#4	0	Automatic calculation for controlled-point shifting
19665#5	0	Controlled-point shift
19666	2.0	Tool holder offset value
19667	X0.0 Y0.0 Z0.0	Controlled-point shift vector
19680	12	Mechanical unit type
19681	4(A)	Controlled axis number for the first rotation axis
19682	1(X)	Axis direction of the first rotation axis
19684	1	Rotation direction of the first rotation axis
19685	0.0	Rotation angle when the first rotation axis is a hypothetical axis
19686	5(B)	Controlled axis number for the second rotation axis

Parameter No.	Setting example	Description
19687	2(Y)	Axis direction of the second rotation axis
19689	1	Rotation direction of the second rotation axis
19690	0.0	Rotation angle when the second rotation axis is a hypothetical axis
19696#0	0	The first rotation axis is an ordinary rotation axis.
19696#1	0	The second rotation axis is an ordinary rotation axis.
19697	3	Direction of the reference tool axis
19700	-10.0	Rotary table position (X-axis)
19701	-30.0	Rotary table position (Y-axis)
19702	0.0	Rotary table position (Z-axis)
19703	0.0	Intersection offset vector between the first and second rotation axes of the table (X-axis)
19704	8.0	Intersection offset vector between the first and second rotation axes of the table (Y-axis)
19705	8.0	Intersection offset vector between the first and second rotation axes of the table (Z-axis)
19709	0.0	Intersection offset vector between the tool axis and the first rotation axis of the tool (X-axis)
19710	0.0	Intersection offset vector between the tool axis and the first rotation axis of the tool (Y-axis)
19711	0.0	Intersection offset vector between the tool axis and the first rotation axis of the tool (Z-axis)
19712	0.0	Intersection offset vector between the second and first rotation axes of the tool (X-axis)
19713	0.0	Intersection offset vector between the second and first rotation axes of the tool (Y-axis)
19714	0.0	Intersection offset vector between the second and first rotation axes of the tool (Z-axis)

⚠ CAUTION (Machine configurations resulting in an alarm)

In the machine settings shown below, there may be no angle solution when G53.1 is issued.
So, specifying G53.1 results in an alarm.

<1>The first rotation axis (master) and second rotation axis (slave) of the tool are on the same axis.
Example: Parameter No. 19682 = parameter No. 19687= 1 (X)

<2>The tool axis direction and the first rotation axis (master) of the table are in the same axis direction.
Example: Parameter No. 19697 = parameter No. 19682 = 3 (Z)

NOTE

1 By setting bit 0 (NSC) of parameter No. 19605 to 1, control point shifting in the tilted working plane indexing of table rotation type can be disabled.

2 The alarm PS5459, "MACHINE PARAMETER INCORRECT" is issued, when tool axis direction control (G53.1/G53.6) is specified on the following condition that parameter is set incorrectly by setting bit 7 (AIR) of parameter No.11221 to 1.
- In case that the movement range of rotation axis (parameter No.19741, 19742, 19743, 19744) is set to the roll-over axis (bit 0 (ROAx) of parameter No.1008 is 1)
- In case that roll-over is set effective (bit 0 (ROAx) of parameter No.1008 is set to 1) for the rotary axis (B type) (bit 0 (ROTx) of parameter No.1006 is set to 1 and bit 1 (ROSx) of parameter No.1006 is set to 1)

Second Rotation Axis Control in Tool Axis Direction Control Where the End Point is a Singular Point

When the end point of tool axis direction control (G53.1/G53.6) during execution of the tilted working plane indexing is a singular point, the second rotation axis is controlled so that the direction of the second feature coordinate system matches that of the workpiece coordinate system.
To enable this function, set bit 4 (CFW) of parameter No. 11221 to 1.

When bit 4 (CFW) of parameter No. 11221 is se to 0, if the end point of tool axis direction control (G53.1/G53.6) is a singular point, the second rotation axis does not operate and only the first rotation axis turns. As a result, the X- and Y-directions of the feature coordinate system depend on the second rotation axis immediately before tool axis direction control. (Fig. 12.16.3.1 (g), Fig. 12.16.3.1 (h))

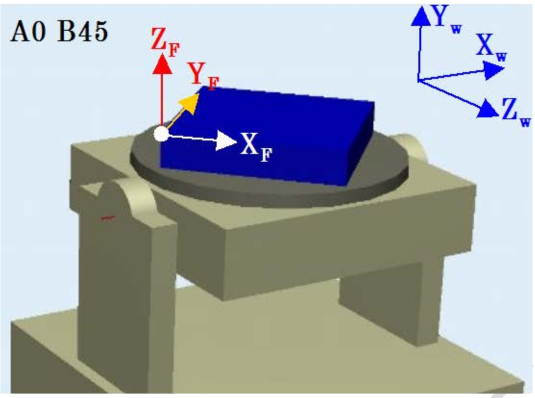


Fig. 12.16.3.1 (g) Before G53.1

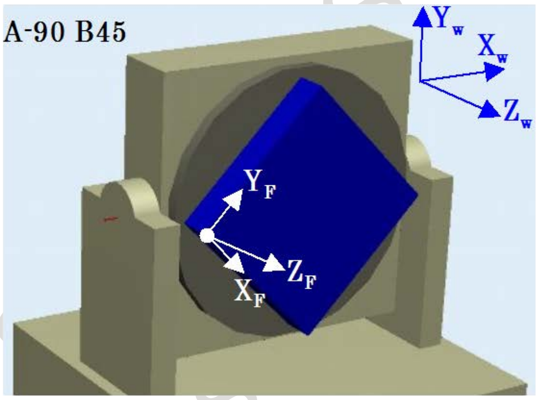


Fig. 12.16.3.1 (h) After G53.1
The second rotation axis (B-axis) does not turn.

When this function is enabled (bit 4 (CFW) of parameter No. 11221 = 1), the second rotation axis is controlled so that the direction of the second feature coordinate system matches that of the workpiece coordinate system. (Fig. 12.16.3.1 (i), Fig. 12.16.3.1 (j))

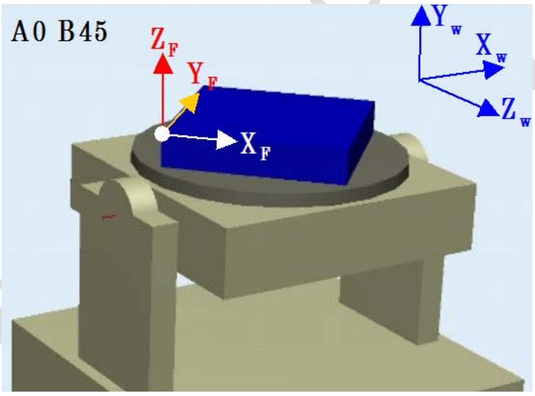


Fig. 12.16.3.1 (i) Before G53.1

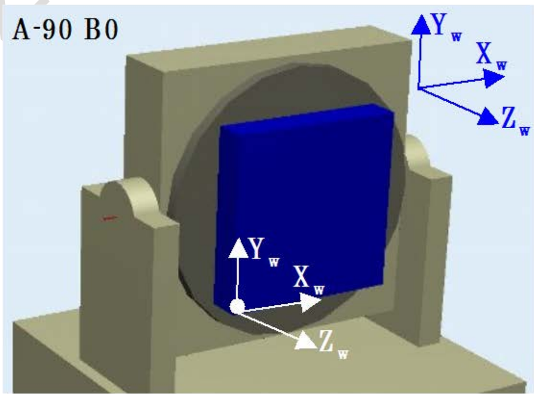


Fig. 12.16.3.1 (j) After G53.1
The second rotation axis (B-axis) moves so that the direction of the second feature coordinate system matches that of the workpiece coordinate system.

12.16.3.2 Tool center point retention type tool axis direction control

In tilted working plane indexing, tool center point retention type tool axis direction control (G53.6) can be specified so that tool can be perpendicular to the tilted plane with the tool center point maintained on the workpiece. In tool center point retention type tool axis direction control, end point of tool center point can be specified. Then tool center point moves on the feature coordinate system fixed on the workpiece. The cycle time can be shortened by moving tool axis direction and tool center point simultaneously.
By specifying the distance between the tool center point to the rotation center with R, it is possible to move the tool so that it is perpendicular to the titled plane while retaining the rotation center shifted from the tool center point.

Explanation

Fig. 12.16.3.2 (a) and Fig. 12.16.3.2 (b) show tool center point retention type tool axis direction control with R not specified. The tool moves so that it is perpendicular to the tilted plane while retaining the tool center point on the workpiece.
This function can be used by specifying G53.6 and specifying a tool length offset number with H (D for the T series). (If H (D) has the modal information for the currently used tool, this function can be used without specifying H (D).)

```
O0002(TCP-HOLD-TYPE)
G00 B0 C0
G5.1 Q1
G68.2 X0 Y0 Z0 I90.0 J45.0 K0
G53.6 H1 (G53.6 D1 for the T series)
```

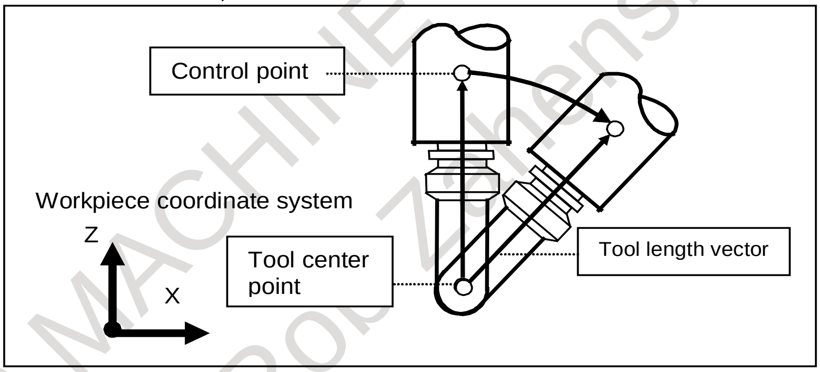


Fig. 12.16.3.2 (a) Operation of tool center point retention type tool axis direction control (tool rotation type)

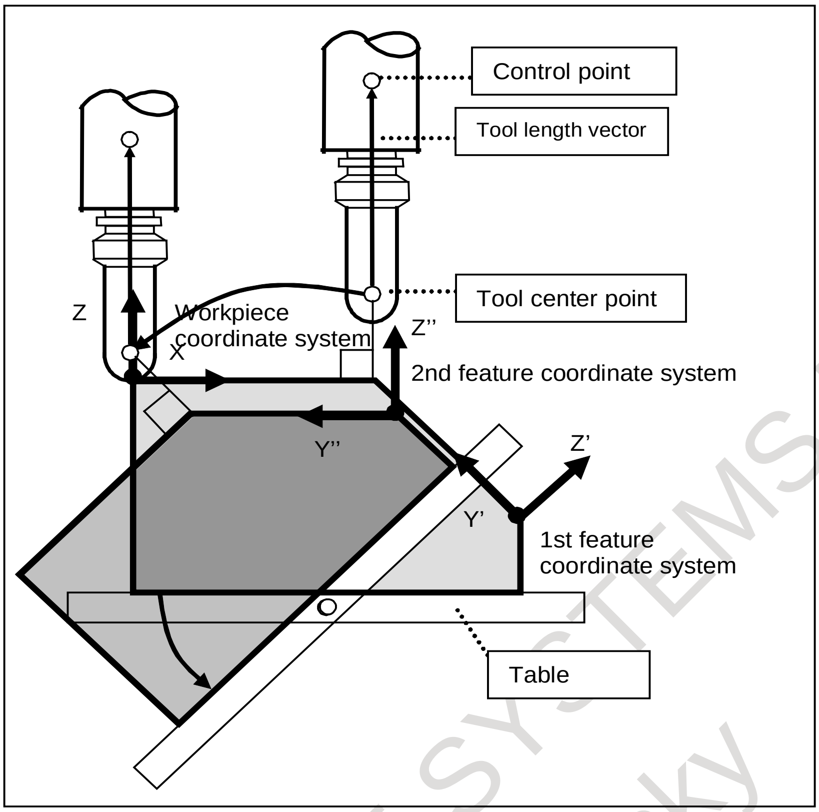


Fig. 12.16.3.2 (b) Operation of tool center point retention type tool axis direction control (table rotation type)

Fig. 12.16.3.2 (c) and Fig. 12.16.3.2 (f) show tool center point retention type tool axis direction control specified with end point of tool center point. Tool center point moves on the feature coordinate system fixed on the workpiece.

O0012(TCP-HOLD-TYPE-TOOL_ROT)
G00 B0 C0
G5.1 Q1
G68.2 X0 Y0 Z0 I90.0 J45.0 K-90.0
G53.6 H1 X100.0 Y0 Z0 (G53.6 D1 X100.0 Y0 Z0 for the T series)

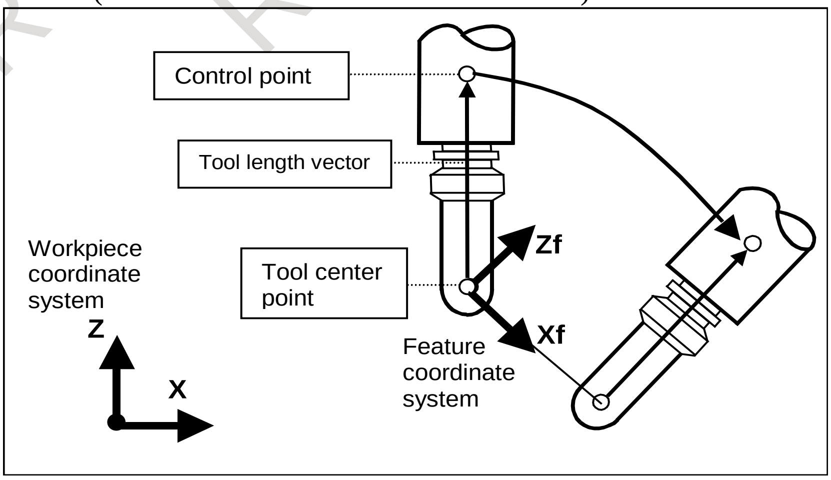


Fig. 12.16.3.2 (c) Operation of tool center point retention type tool axis direction control specified with end point of tool center point (tool rotation type)

O0022(TCP-HOLD-TYPE-TABLE_ROT)
G00 B0 C0

```
G5.1 Q1
G68.2 X0 Y0 Z0 I90.0 J45.0 K-90.0
G53.6 H1 X0 Y0 Z0 (G53.6 D1 X0 Y0 Z0 for the T series)
```

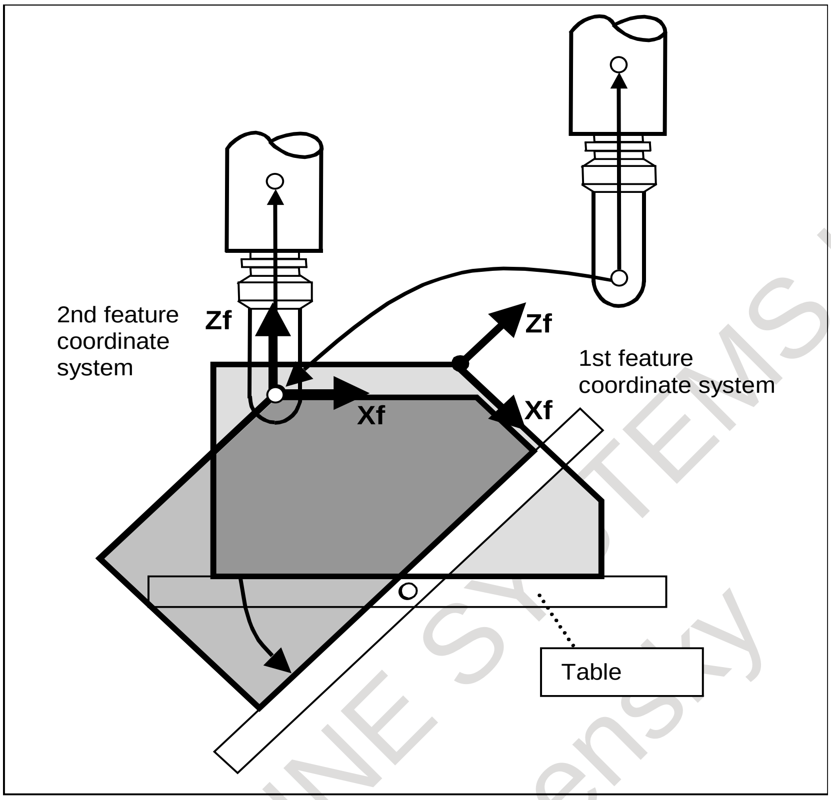


Fig. 12.16.3.2 (d) Operation of tool center point retention type tool axis direction control specified with end point of tool center point (table rotation type)

Rotation center compensation with the tool center point retention type

In tool center point retention type tool axis direction control, the rotation center can be shifted from the tool center point by specifying R.
Fig. 12.16.3.2 (e) and Fig. 12.16.3.2 (f) show cases in which the rotation center is shifted by specifying the distance from the tool center point to the workpiece with R.
By specifying this, the tool moves so that it is perpendicular to the titled plane while retaining the rotation center on the workpiece.
This function can be used by specifying G53.6 and specifying a tool length offset number with H (D for the T series) and specifying the distance from the tool center point to the rotation center with R.

```
O0003(CENTER-OF-ROTATION-HOLD-TYPE)
G00 B0 C0
G5.1 Q1
G68.2 X0 Y0 Z0 I90.0 J45.0 K0
G53.6 H1 R200.0 (G53.6 D1 R200.0 for the T series)
```

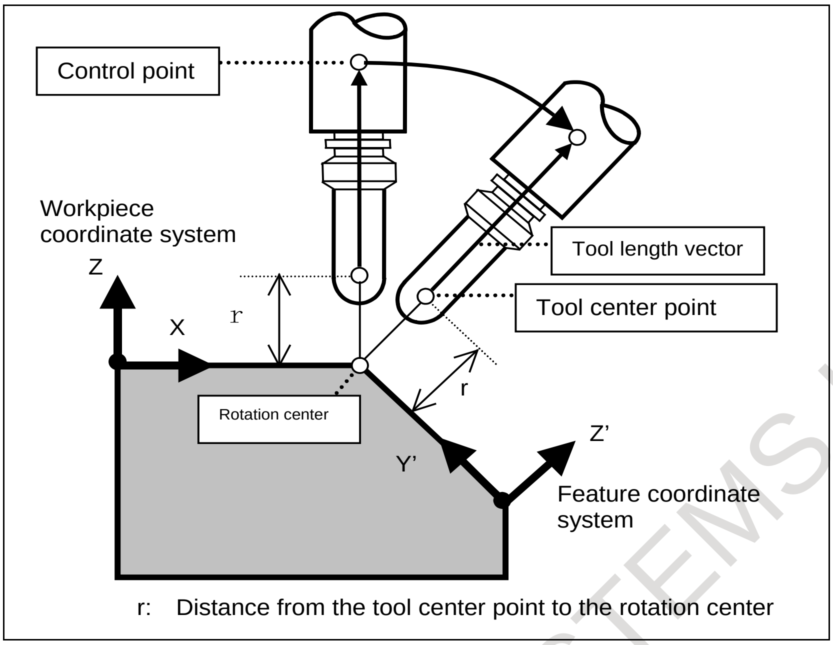


Fig. 12.16.3.2 (e) Operation of rotation center compensation with tool center point retention type tool axis direction control (tool rotation type)

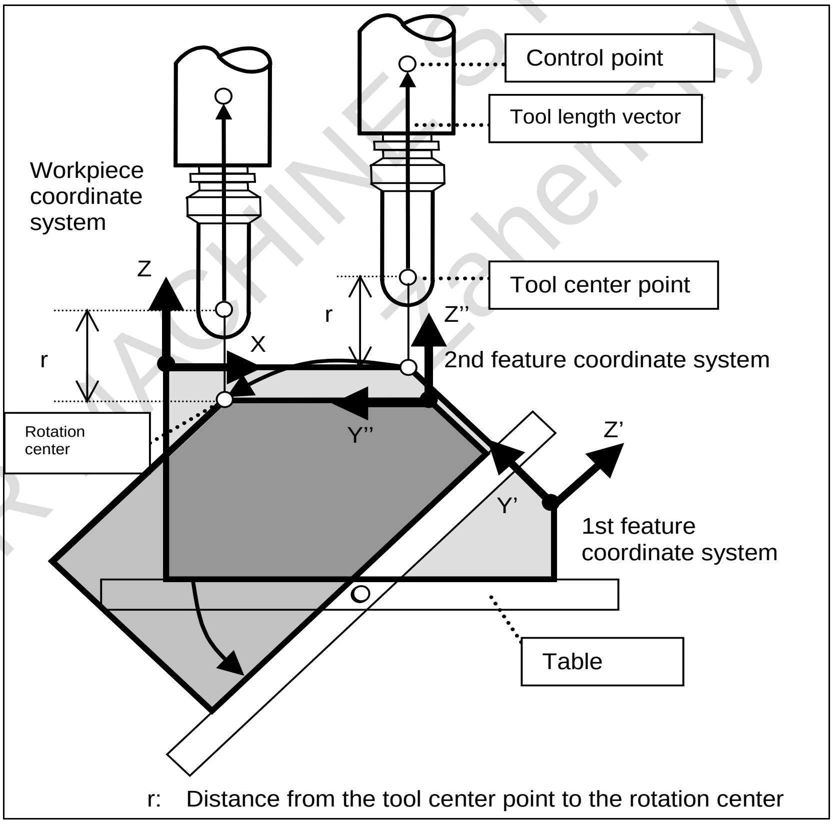


Fig. 12.16.3.2 (f) Operation of rotation center compensation with tool center point retention type tool axis direction control (table rotation type)

12.16.4 Tilted Working Plane Indexing in Tool Length Compensation

Overview

In tool length compensation (G43), G68.2/G68.4 (tilted working plane indexing) and G53.1 (tool axis direction control)/G53.6 (tool center point retention type tool axis direction control) can be specified.
Accordingly, the G68.2/G68.4 and G53.1/G53.6 commands can be used without canceling tool length compensation.

Explanation

- G68.2/G68.4 command in tool length compensation

The G68.2/G68.4 command can be executed in tool length compensation.
Absolute coordinates after the G68.2/G68.4 command are based on the position of the tool center point on the feature coordinate system.
When the tilted working plane indexing is executed with the tool or table tilted on the rotation axis, absolute coordinates are based on the position of the tool center point with the position of the rotation axis considered.
Accordingly, machining is allowed even when the tool axis direction is not the Z-axis direction on the feature coordinate system.

Example of operation 1

N10 G69 ;
N20 G54 **G43** H1 X0 Y0 Z0 ;
N30 **G68.2** X_ Y_ Z_ I90. 0J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N40 X100.0 Y0 Z0 ;

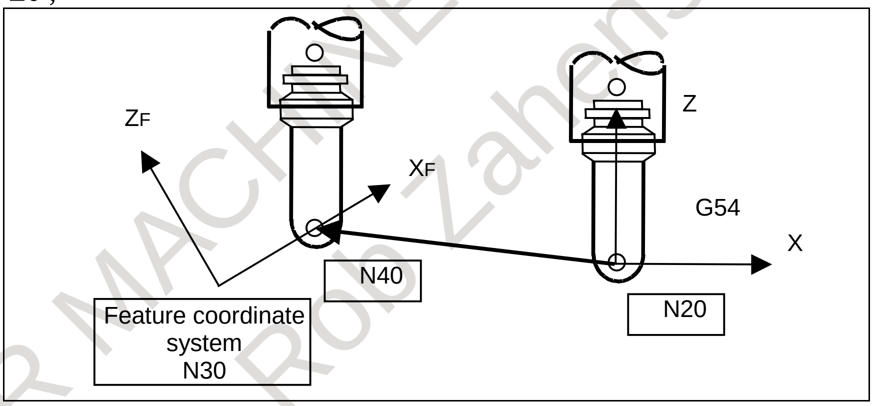


Fig. 12.16.4 (a) Example of operation 1

Example of operation 2

N10 G69 ;
N20 G54 **G43** H1 X0 Y0 Z0 B0 ;
N30 B20.0 ;
N40 **G68.2** X_ Y_ Z_ I90.0 J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N50 X100.0 Y0 Z0 ;

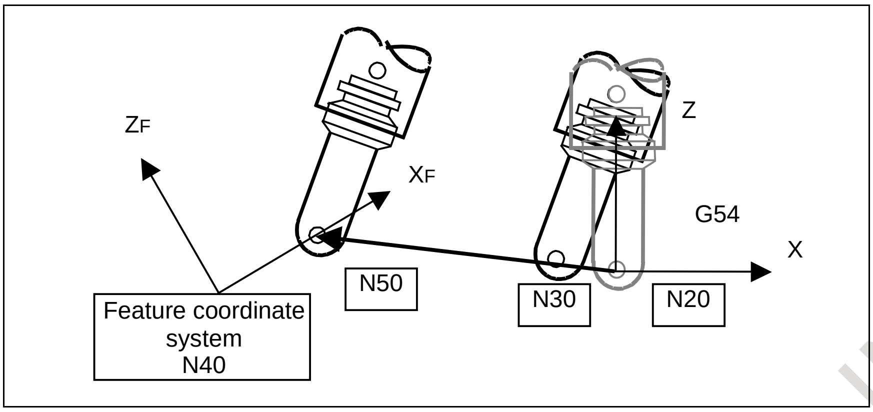


Fig. 12.16.4 (b) Example of operation 2 (tool rotation type)

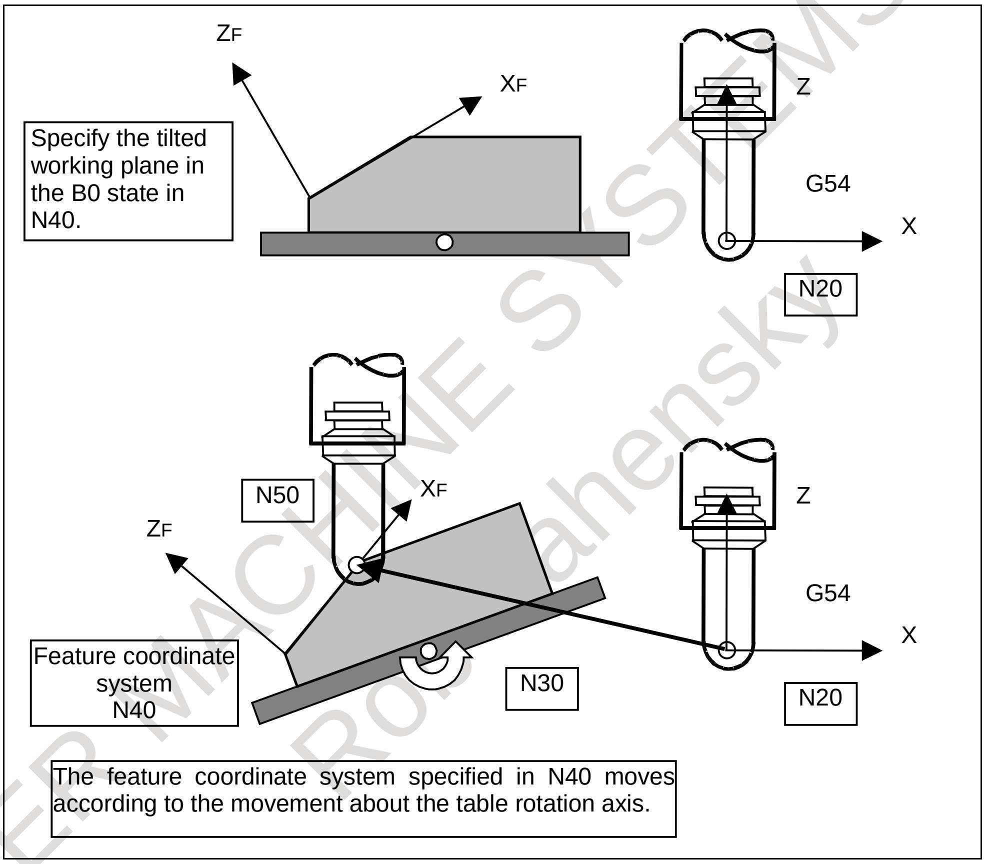


Fig. 12.16.4 (c) Example of operation 2 (table rotation type)

Example of operation 3

N30 **G43** X0 Y0 Z0 B20.0 ;
N40 **G68.2** X_ Y_ Z_ I90.0 J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N50 X100.0 Y0 Z0 ;
N60 B-20.0
N70 **G68.4** X_ Y_ Z_ I90.0 J40.0 K-90.0 ;（incremental multiple command: rotation by 40 degrees about the Y-axis）
N80 X100.0 Y0 Z0 ;

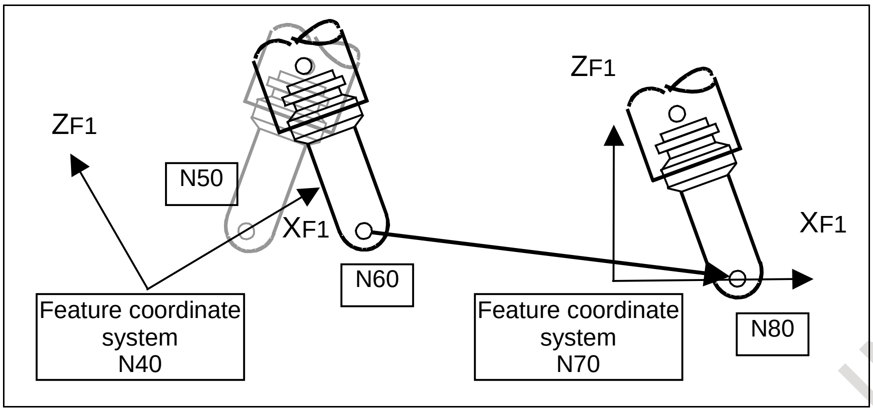


Fig. 12.16.4 (d) Example of operation 3 (tool rotation type)

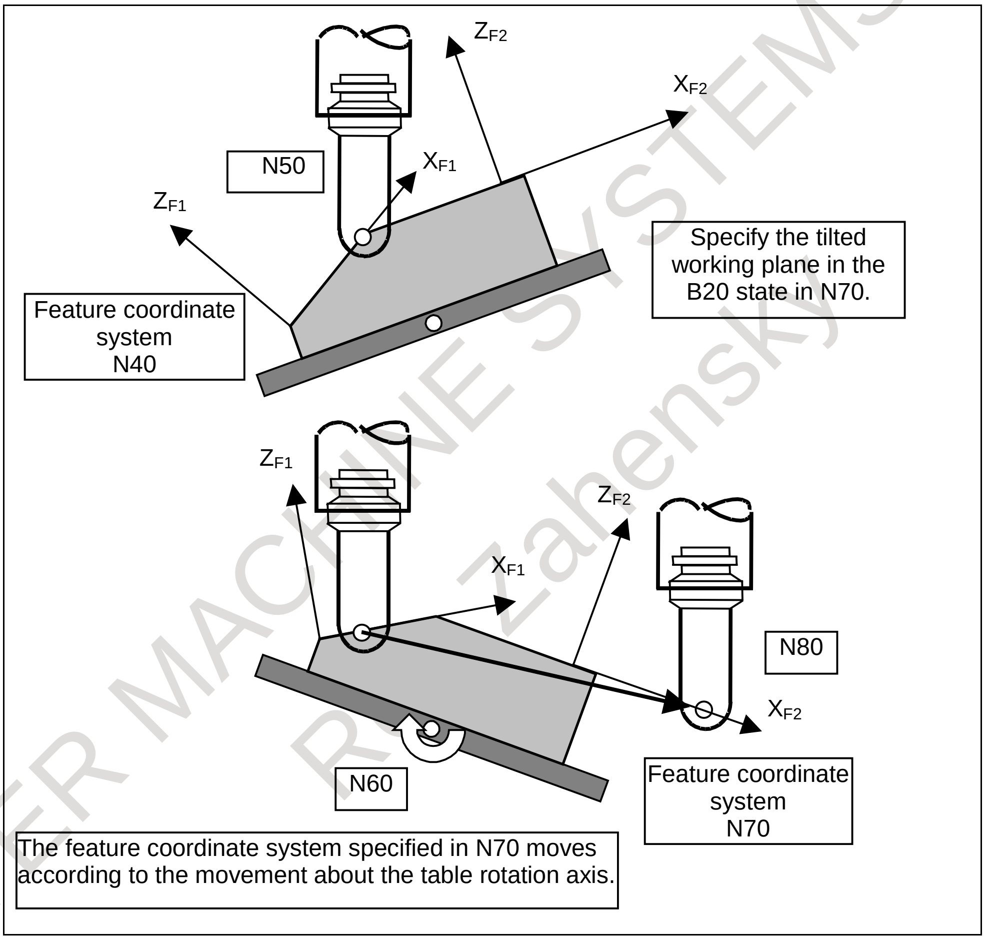


Fig. 12.16.4 (e) Example of operation 3 (table rotation type)

G53.1 command in tool length compensation

The G53.1 command can be executed in tool length compensation.

G53.1 operation in tool length compensation is performed in the same way as in the tool length compensation cancel mode.

Absolute coordinates after the G53.1 command are based on the position of the tool center point on the feature coordinate system after the G53.1 command is specified.

Example of operation 4

N10 G54 **G43** H1 X_ Y_ Z_ ;
N20 **G68.2** X_ Y_ Z_ I90.0 J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N30 **G53.1** ;
N40 X100.0 Y0 Z0 ;

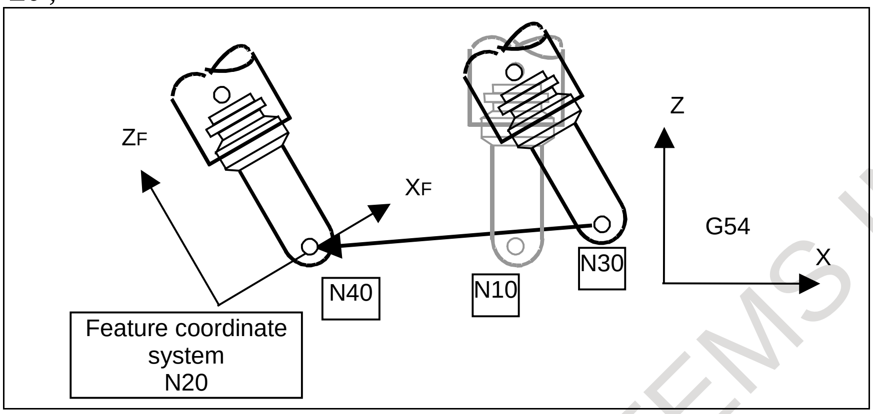


Fig. 12.16.4 (f) Example of operation 4 (tool rotation type)

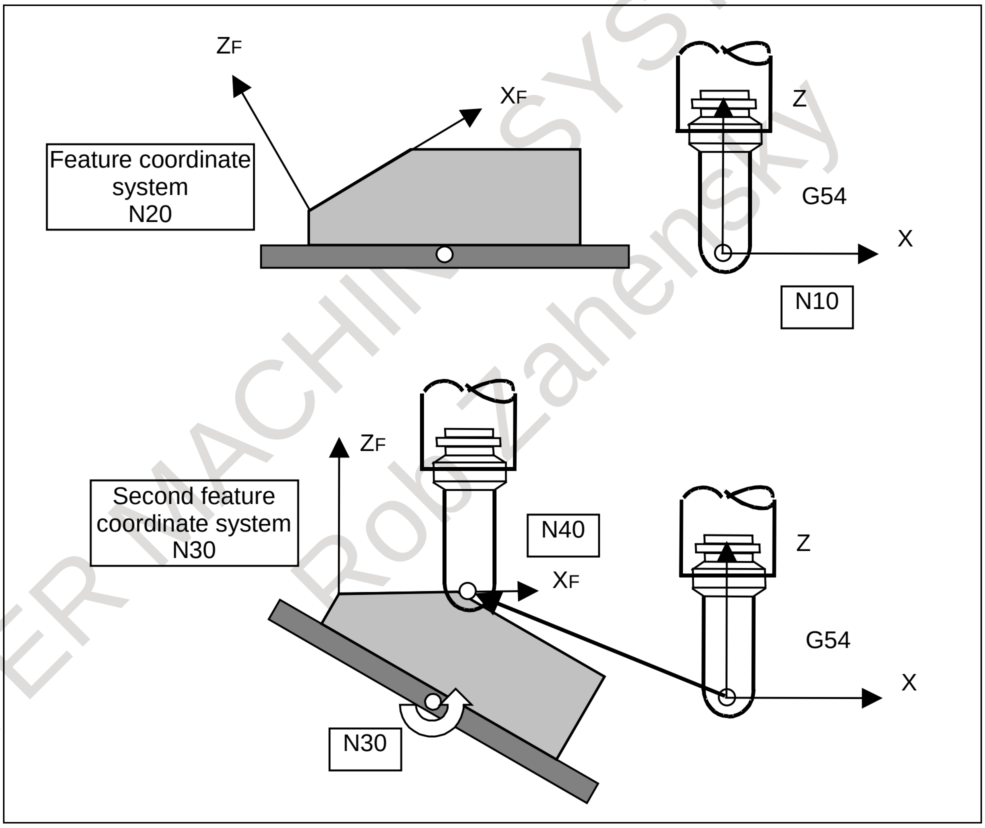


Fig. 12.16.4 (g) Example of operation 4 (table rotation type)

G53.6 command in tool length compensation

The G53.6 command can be executed in tool length compensation.
G53.6 operation in tool length compensation is performed in the same way as in the tool length compensation cancel mode.

G69 command in tool length compensation

The G69 command can be executed in tool length compensation.
After G69 operation in tool length compensation, the tool length compensation vector represents the Z direction of the workpiece coordinate system regardless of the position of the rotation axis.

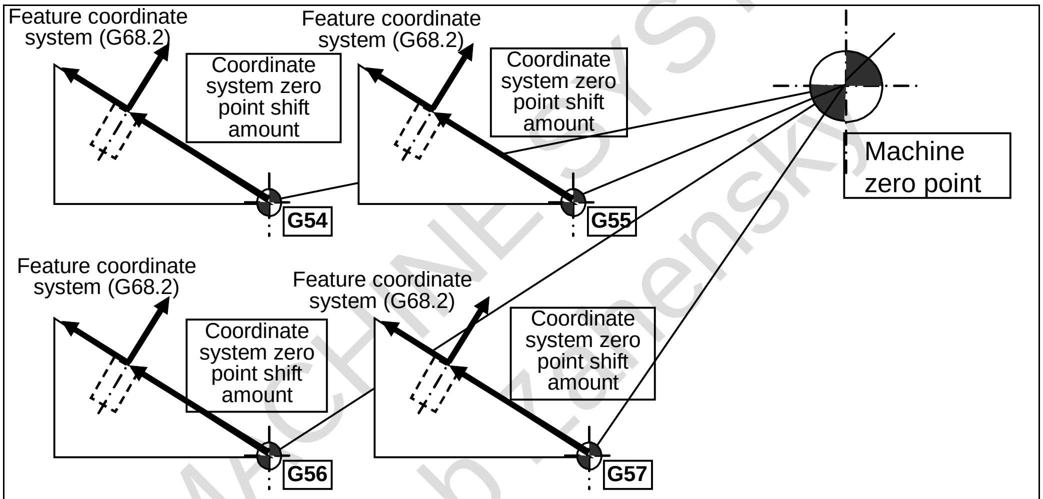
Parameter

		#7	#6	#5	#4	#3	#2	#1	#0
1205			3TW						

[Input type] Parameter input
[Data type] Bit path

#6 3TW When workpiece coordinate system selection is specified with G code in tilted working plane indexing mode:
0: The alarm PS5462, "ILLEGAL COMMAND (G68.2/G69)" is issued.
1: Workpiece coordinate system selection is executed.

> ⚠ **CAUTION**
> When this parameter is 1, only G54 to G59 or G54.1 can be specified. Specifying G52 or G92 causes alarm PS5462. Specifying G54 to G59 or G54.1 suppresses buffering.



Program example
G90 G17
G54
G68.2 X_Y_Z_ I_ J_ K_
G53.1
G43H_
G55 X_Y_Z_
G56 X_Y_Z_
G57 X_Y_Z_
G49
G69

X_Y_Z_: Coordinate system zero point shift amount

		#7	#6	#5	#4	#3	#2	#1	#0
3106			DAK						

[Input type] Setting input
[Data type] Bit

#6 DAK Specifies whether to display coordinates in the program coordinate system or workpiece coordinate system as absolute coordinates when the 3-dimensional coordinate conversion mode, the tilted working plane indexing mode is set.
0: Display coordinates in the program coordinate system.

1: Display coordinates in the workpiece coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
5400			LV3					

[Input type] Parameter input
[Data type] Bit path

#5 LV3 When system variables #100101 to #100132 (current position coordinates) and #100151 to #100182 (skip coordinates) are read in the 3-dimensional coordinate system conversion mode or tilted working plane indexing mode:

0: Coordinates of the workpiece coordinate system can be read.

1: Coordinates of the program coordinate system after 3-dimensional coordinate system conversion or tilted working plane indexing can be read.

This parameter applies also to system variables #5041 to #5060 (current position coordinates) and #5061 to #5080 (skip coordinates).

5412	Rapid traverse rate for canned cycle for drilling in 3-dimensional coordinate system conversion mode

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets a rapid traverse rate for canned cycle for drilling in the 3-dimensional coordinate system conversion mode, the tilted working plane indexing mode.

	#7	#6	#5	#4	#3	#2	#1	#0
11216								D3O

[Input type] Parameter input
[Data type] Bit path

#0 D3O When the bit 1 (D3M) of parameter No.11600 set to "1" and Override is 0%, during Tilted working plane indexing or 3-dimensional coordinate system conversion mode, the axis moving signals "MV1 to MV8<Fn102>" :

- 3-dimensional coordinate system conversion
- Tilted working plane indexing

0: becomes 0.

1: maintains the state of "1".

> **NOTE**
> When the manual operation (Jog feed / Incremental feed / Manual handle interrupt)is being executed, the bit 0 (D3O) of parameter No.11216 become invalid. It becomes operation of corresponding D3O=0.

11220	Minimum distance used for determining a plane when a tilted working plane indexing with three points is specified

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (input unit)

[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

When a tilted working plane indexing with three points is specified, if the distance (used for determining a place) between a straight line passing two points and the remaining one point is short, the plane is unstable. In this parameter, set the minimum distance used for determining a plane. If the distance is shorter than the value set in this parameter, an alarm PS5457, “G68.2/G68.3 FORMAT ERROR” is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
11221	AIR			CFW			D3R	MTW

[Input type] Parameter input

[Data type] Bit path

#0 MTW Multiple commands of tilted working plane indexing are:

0: Not used.

1: Used.

#1 D3R In the 3-dimensional coordinate system conversion mode, tilted working plane indexing mode, rapid traverse in canned cycle for drilling is:

0: Performed in the cutting feed mode.

1: Performed in the rapid traverse mode.

#4 CFW When the end point of tool axis direction control (G53.1/G53.6) during execution of the tilted working plane indexing is a singular point:

0: The second rotation axis is not operated.

1: The second rotation axis is controlled so that the direction of the second feature coordinate system matches that of the workpiece coordinate system.

#7 AIR If the movement range of rotation axis (parameter No.19741, 19742, 19743, 19744) is set to the roll-over axis, or roll-over function is set to rotary axis (B type), when tool axis direction control is executed:

0: The alarm PS5459, "MACHINE PARAMETER INCORRECT" is not issued.

1: The alarm PS5459, "MACHINE PARAMETER INCORRECT" is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
11304							GGD	

[Input type] Parameter input

[Data type] Bit

NOTE

When this parameter is set, the power must be turned off before operation is continued.

#1 GGD The G code guidance screen is:

0: Not displayed.

1: Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
11600							D3MV	

[Input type] Parameter input
[Data type] Bit path

#1 D3MV In following modes, axis moving signals MV1-MV8 <Fn0102> is:
- 3-dimensional coordinate system conversion
- Tilted working plane indexing

0: The signals for axes on programming coordinate system.
1: The signals for axes on workpiece coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
11630						TFR		

[Input type] Parameter input
[Data type] Bit path

#2 TFR The minimum command unit of the rotation angles of the tilted working plane indexing is:
0: 0.001 degree.
1: 0.00001 degree.

	#7	#6	#5	#4	#3	#2	#1	#0
13451							ATW	

[Input type] Parameter input
[Data type] Bit path

#1 ATW When I, J, and K are all set to 0 in a block that specifies a feature coordinate system setup command (G68.2), which is a tilted working plane indexing:
0: An alarm PS5457 "G68.2/G68.3 FORMAT ERROR" is issued.
1: A feature coordinate system with a tilted plane angle of 0 degrees is assumed for operation.

	#7	#6	#5	#4	#3	#2	#1	#0
19602			D3D					

[Input type] Parameter input
[Data type] Bit path

#5 D3D Specifies whether to display the distance to go in the program coordinate system or workpiece coordinate system during the 3-dimensional coordinate system conversion or the tilted working plane indexing.
0: Display the distance to go in the program coordinate system.
1: Display the distance to go in the workpiece coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
19605								NSC

[Input type] Parameter input
[Data type] Bit path

#0 NSC For the machine type that has no rotation axis for rotating the tool (when parameter No. 19680 is set to 12 to specify the table rotation type), control point shifting in the tilted working plane indexing is:

0: Enabled.
Set bit 4 (SPR) and bit 5 (SVC) of parameter No. 19665.

1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
19608			PRI					

[Input type] Parameter input
[Data type] Bit path

#5 PRI Among multiple end point candidates that exist when a movement is made on a rotation axis by a command such as I, J, and K when a slanted surface machining command is specified under 3-dimensional tool compensation (type 2):

0: A combination in which the master (first rotation axis) makes a smaller angular movement is selected for a machine of tool rotation type or table rotation type. A combination in which the table (second rotation axis) makes a smaller angular movement is selected for a machine of composite type.

1: A combination in which the slave (second rotation axis) makes a smaller angular movement is selected for a machine of tool rotation type or table rotation type. A combination in which the tool (first rotation axis) makes a smaller angular movement is selected for a machine of composite type.

	#7	#6	#5	#4	#3	#2	#1	#0
19665			SVC	SPR				

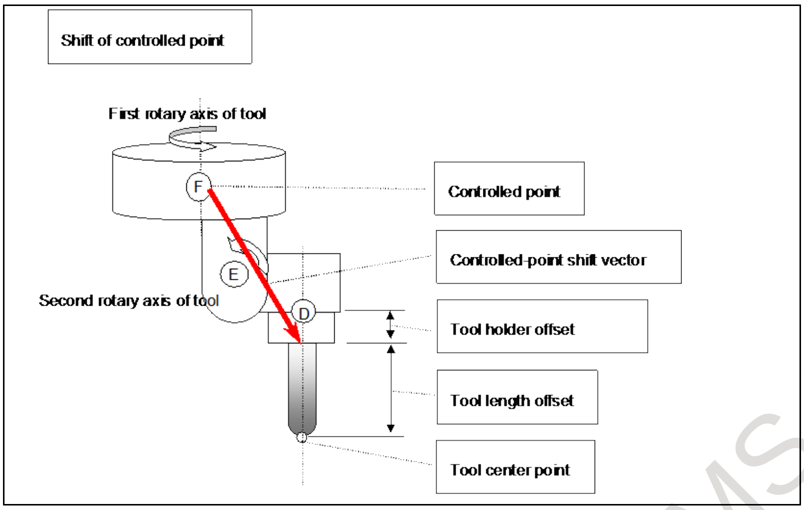
[Input type] Parameter input
[Data type] Bit path

#4 SPR The controlled point is shifted by:

0: Automatic calculation.

1: Using parameter No. 19667.

Bit 5 (SVC) of parameter No. 19665	Bit 4 (SPR) of parameter No. 19665	Shift of controlled point
0	-	Shift is not performed as not done conventionally.
1	0	The controlled point is shifted according to the result of the following automatic calculation: - (Intersection offset vector between the tool axis and the first rotation axis of the tool + intersection offset vector between the second and first rotation axes of the tool + tool holder offset (parameter No. 19666)) (See the figure next.)
1	1	The controlled point is shifted. As the shift vector, the vector set in parameter No. 19667 is used.



[Controlled-point shift vector when automatically calculated]

#5 SVC The controlled point is:
0: Not shifted.
1: Shifted.
The method of shifting is specified by bit 4 (SPR) of parameter No. 19665.

> **NOTE**
> When the machine has no rotation axis for rotating the tool (when parameter No. 19680 is set to 12 to specify the table rotation type), the controlled point is not shifted regardless of the setting of this parameter.

19666	Tool holder offset value

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set an offset value (tool holder offset value) specific to the machine from the control point to the tool attachment position in tool length compensation (after specification of G53.1 in the tilted working plane indexing mode). In tool length compensation (not in the tilted working plane indexing mode), however, tool holder offset can be enabled or disabled with bit 7 (ETH) of parameter No. 19665.

> **NOTE**
> Set a radius value.

19667	Controlled-point shift vector

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the shift vector for the controlled point. This value becomes valid when bit 5 (SVC) of parameter No. 19665 is set to 1, and bit 4 (SPR) of parameter No. 19665 is set to 1.

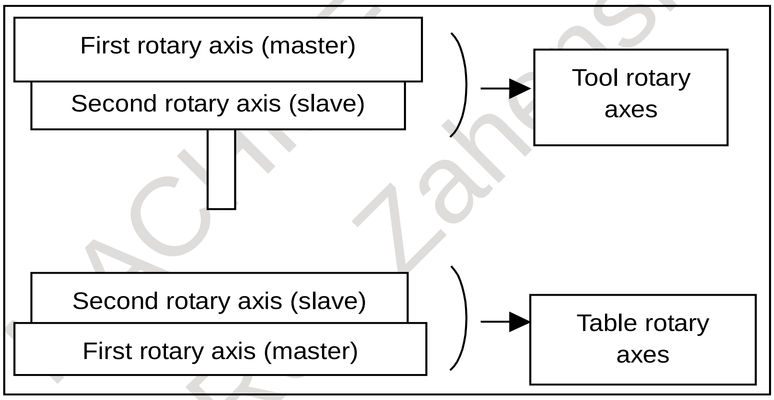
> **NOTE**
> Set a radius value.

19680	Mechanical unit type

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 21

Specify the type of the mechanical unit.

Parameter No. 19680	Mechanical unit type	Controlled rotation axis	Master and slave
0		Mechanism having no rotation axis	
2	Tool rotation type	Two rotation axes of the tool	The first rotation axis is the master, and the second rotation axis is the slave.
12	Table rotation type	Two rotation axes of the table	The first rotation axis is the master, and the second rotation axis is the slave.
21	Mixed type	One rotation axis of the tool + one rotation axis of the table	The first rotation axis is the tool rotation axis, and the second rotation axis is the table rotation axis.



> **NOTE**
> A hypothetical axis is also counted as a controlled rotary axis.
> <Hypothetical axis>
> In some cases, it is convenient to use an imaginary rotary axis whose angle is fixed to a certain value. For example, suppose that a tool is mounted in a tilted manner through an attachment. In such a case, the rotary axis considered hypothetically is a hypothetical axis. Bits 0 (IA1) and 1 (IA2) of parameter No. 19696 determine whether each rotary axis is an ordinary rotary axis or a hypothetical axis.

19681	Controlled-axis number for the first rotation axis

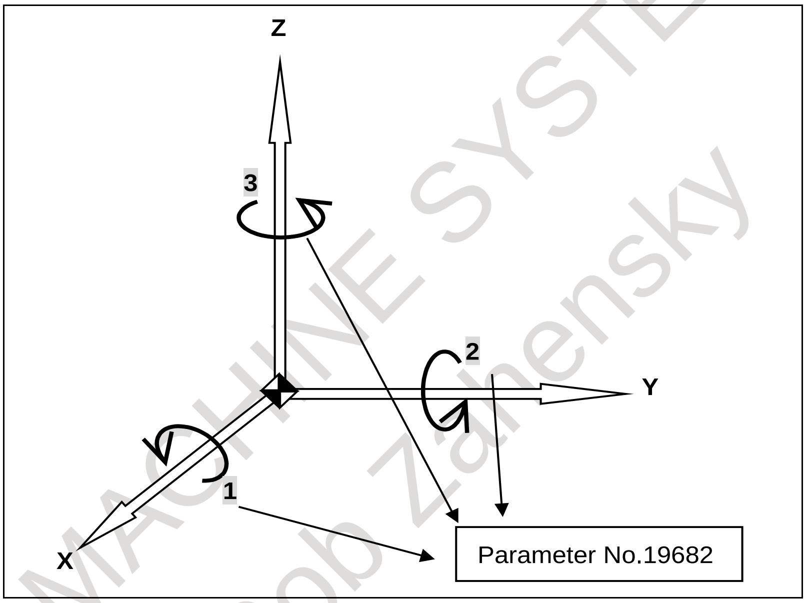
[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the controlled-axis number for the first rotation axis.
For a hypothetical axis (when bit 0 (IA1) of parameter No. 19696 is 1), set 0.
Example)
Assuming that the axis configuration in path 1 is X,Y,Z,B,C and the axis configuration in path 2 is X,Z,C,Y,B, set the parameter to 5 in path 1 and to 3 in path 2 if C is the first rotation axis in each path.

19682	Axis direction of the first rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

Specify the axis direction of the first rotation axis.
1: On X-axis
2: On Y-axis
3: On Z-axis



19684	Rotation direction of the first rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 1

Set the direction in which the first rotation axis rotates as a mechanical motion when a positive move command is issued.
0: Clockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19682 (right-hand thread rotation)
1: Counterclockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19682 (left-hand thread rotation)

Normally, 0 is set for a tool rotation axis, and 1 is set for a table rotation axis.

19685	Rotation angle when the first rotation axis is a hypothetical axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
When the first rotation axis is a hypothetical axis (bit 0 (IA1) of parameter No. 19696 is 1), set the rotation angle.

19686	Controlled-axis number for the second rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the controlled-axis number for the second rotation axis.
For a hypothetical axis (bit 1 (IA2) of parameter No. 19696 is 1), set 0.
[Example] Assuming that the axis configuration in path 1 is X,Y,Z,B,C and the axis configuration in path 2 is X,Z,C,Y,B, set the parameter to 4 in path 1 and to 5 in path 2 if B is the second rotation axis in each path.

19687	Axis direction of the second rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3
Specify the axis direction of the second rotation axis.
1: On X-axis
2: On Y-axis
3: On Z-axis
When the second rotation axis is the slave axis, the direction when the master axis is at 0 degrees must be set.

19689	Rotation direction of the second rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 1
Set the direction in which the second rotation axis rotates as a mechanical motion when a positive move command is issued.
0: Clockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19687 (right-hand thread rotation)
1: Counterclockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19687 (left-hand thread rotation)
Normally, 0 is set for a tool rotation axis, and 1 is set for a table rotation axis.

19690	Rotation angle when the second rotation axis is a hypothetical axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
When the second rotation axis is a hypothetical axis (bit 1 (IA2) of parameter No. 19696 is 1), set the rotation angle.

	#7	#6	#5	#4	#3	#2	#1	#0
19696							IA2	IA1

[Input type] Parameter input
[Data type] Bit path

#0 IA1

0: The first rotation axis is an ordinary rotation axis.
1: The first rotation axis is a hypothetical axis.

If IA1 is 1, set 0 as the controlled-axis number for the first rotation axis (parameter No. 19681).
Also, set parameters Nos. 19682 to 19685 on the assumption that there is a rotation axis.

#1 IA2

0: The second rotation axis is an ordinary rotation axis.
1: The second rotation axis is a hypothetical axis.

If IA2 is 1, set 0 as the controlled-axis number for the second rotation axis (parameter No. 19686).
Also, set parameters Nos. 19687 to 19690 on the assumption that there is a rotation axis.

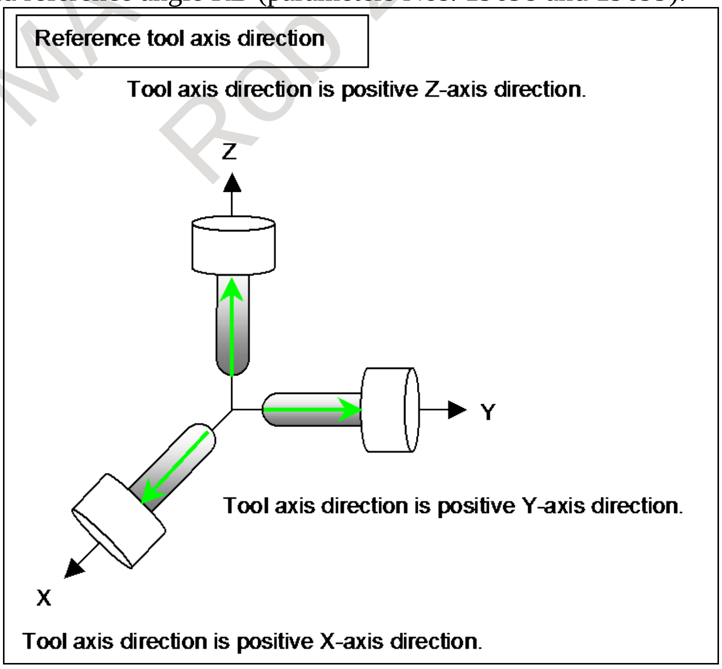
19697	Reference tool axis direction

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

Set the tool axis direction in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees. Also, set the tool axis direction in the machine coordinate system in a mechanism in which only the rotation axes for controlling the table are present (there is no rotation axis for controlling the tool).

1: Positive X-axis direction
2: Positive Y-axis direction
3: Positive Z-axis direction

When the reference tool axis direction is neither the X-, Y-, nor Z-axis direction, set the reference direction in this parameter, then set appropriate angles as the reference angle RA and reference angle RB (parameters Nos. 19698 and 19699).

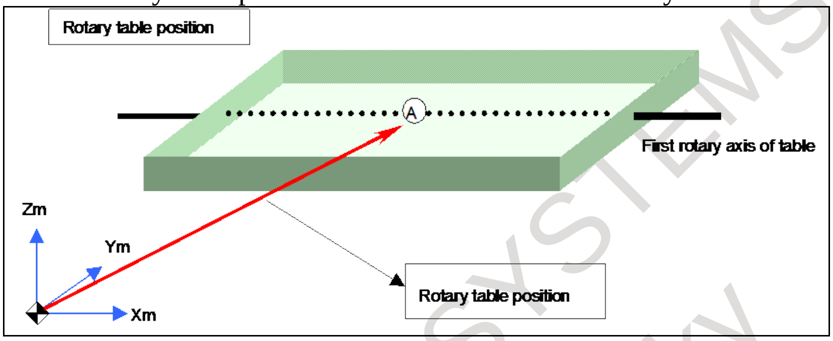


19700	Rotary table position (X-axis of the basic three axes)

19701	**Rotary table position (Y-axis of the basic three axes)**

19702	**Rotary table position (Z-axis of the basic three axes)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set these parameters when parameter No. 19680 is set to 12 or 21. The vector from the origin of the machine coordinate system to point A on the first rotation axis of the table is set as the rotary table position in the machine coordinate system.



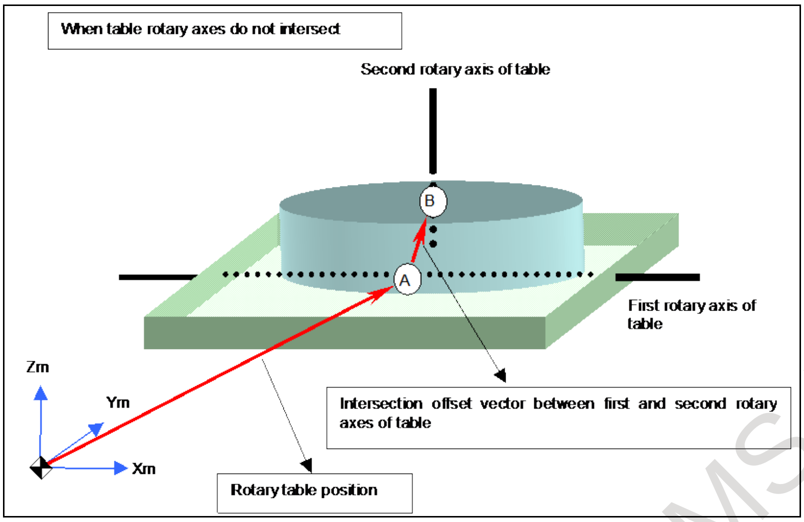
> **NOTE**
> As point A, set a position that is easy to measure on the first rotary axis of the table.
> Set a radius value.
> If the rotary table is moved along the X-, Y-, or Z-axis or all of these axes, set the position of the rotary table when the machine coordinates of the X-, Y-, and Z-axes are all set to 0.

19703	**Intersection offset vector between the first and second rotation axes of the table (X-axis of the basic three axes)**

19704	**Intersection offset vector between the first and second rotation axes of the table (Y-axis of the basic three axes)**

19705	**Intersection offset vector between the first and second rotation axes of the table (Z-axis of the basic three axes)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set these parameters when the first rotation axis and second rotation axis of the table do not intersect. These parameters are valid when parameter No. 19680 is set to 12. When the rotation axes for controlling the table are all at 0 degrees, the vector from point A to point B on the second rotation axis of the table is set as the intersection offset vector in the machine coordinate system.

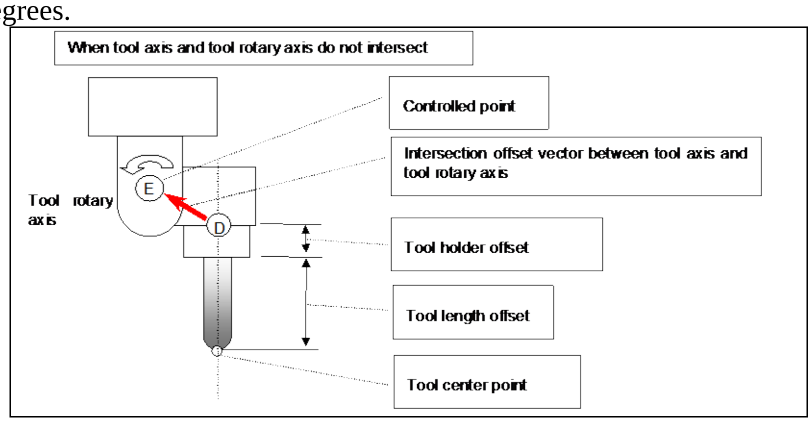


> **NOTE**
> As point B, set a position that is easy to measure on the second rotary axis of the table.
> Set a radius value.

19709	Intersection offset vector between the tool axis and tool rotation axis (X-axis of the basic three axes)
19710	Intersection offset vector between the tool axis and tool rotation axis (Y-axis of the basic three axes)
19711	Intersection offset vector between the tool axis and tool rotation axis (Z-axis of the basic three axes)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

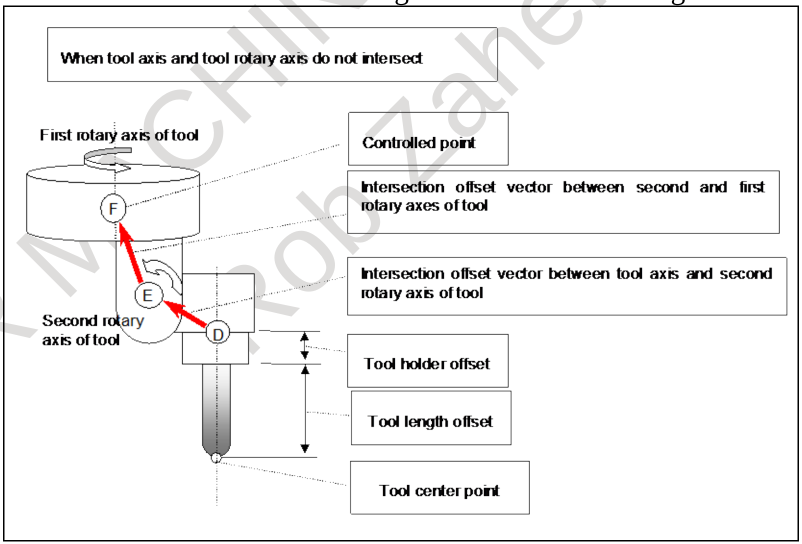
Set these parameters when the tool axis and tool rotation axis do not intersect.
These parameters are valid when parameter No. 19680 is set to 2 or 21.
If parameter No. 19680 is 21, set the vector from point D on the tool axis to point E determined on the tool rotation axis as the intersection offset vector in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees.
If parameter No. 19680 is 2, set the vector from point D on the tool axis to point E determined on the second rotation axis of the tool as the intersection offset vector in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees.



> **NOTE**
> Point D is determined by adding the tool length offset and tool holder offset (parameter No. 19666) to the tool tip. As point E, set a position that is easy to measure.
> Set a radius value.

19712	**Intersection offset vector between the second and first rotation axes of the tool (X-axis of the basic three axes)**
19713	**Intersection offset vector between the second and first rotation axes of the tool (Y-axis of the basic three axes)**
19714	**Intersection offset vector between the second and first rotation axes of the tool (Z-axis of the basic three axes)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set these parameters when the rotation axes of the tool do not intersect.
These parameters are valid when parameter No. 19680 is set to 2.
Set the vector from point E on the second rotation axis of the tool to point F on the first rotation axis of the tool as the intersection offset vector in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees.



> **NOTE**
> As point F, set a position that is easy to measure.
> Set a radius value.

19741	**Upper limit of the movement range of the first rotation axis**

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the upper limit of the movement range of the first rotation axis in tool axis direction control of the tilted working plane indexing (G53.1). When the movement range of the first rotation axis is not specified or the first rotation axis is the roll-over axis, this parameter and parameter No. 19742 must both be set to 0.

19742	Lower limit of the movement range of the first rotation axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the lower limit of the movement range of the first rotation axis in tool axis direction control of the tilted working plane indexing (G53.1). When the movement range of the first rotation axis is not specified or the first rotation axis is the roll-over axis, this parameter and parameter No. 19741 must both be set to 0.

19743	Upper limit of the movement range of the second rotation axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the upper limit of the movement range of the second rotation axis in tool axis direction control of the tilted working plane indexing (G53.1). When the movement range of the second rotation axis is not specified or the second rotation axis is the roll-over axis, this parameter and parameter No. 19744 must both be set to 0.

19744	Lower limit of the movement range of the second rotation axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the lower limit of the movement range of the second rotation axis in tool axis direction control of the tilted working plane indexing (G53.1). When the movement range of the second rotation axis is not specified or the second rotation axis is the roll-over axis, this parameter and parameter No. 19743 must both be set to 0.

12.16.5 Restrictions of Tilted Working Plane Indexing

- Basic restrictions

The restrictions imposed on 3-dimensional coordinate conversion also apply to the tilted working plane indexing.
The following presents the restrictions to bear in mind in particular:

- Increment system

The same increment system must be used for the basic three axes used by this function.

- Rapid traverse command

Linear rapid traverse (bit 1 (LRP) of parameter No. 1401=1) must be set for the rapid traverse command.

- 3-dimensional coordinate system conversion

If an attempt is made to set a new coordinate system by performing 3-dimensional coordinate conversion in a feature coordinate system, an alarm is also raised.

- Positioning in the machine coordinate system

Positioning commands in the machine coordinate system such as G28, G30, and G53 operate in the machine coordinate system rather than the feature coordinate system.

- External mirror image

If an attempt is made to use this function and an external mirror image function simultaneously, this function takes effect before the external mirror image function.

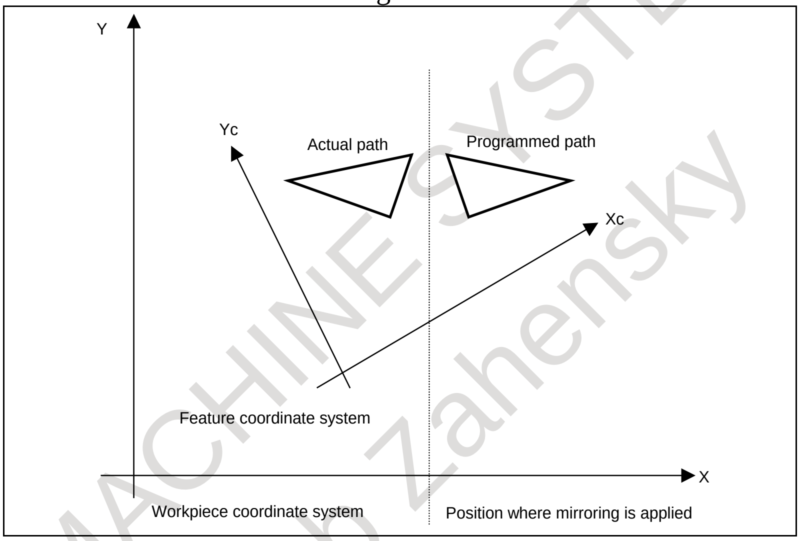


Fig. 12.16.5 (a)

- Tool center point retention type tool axis direction control

Reset

Performing a reset during tool center point retention type tool axis direction control results in the operation to be performed when the following parameters are set regardless of whether the reset is performed in the foreground or background. Thus, do not use the background during tool center point retention type tool axis direction control.

(1) Bit 2 (D3R) of parameter No. 5400 = 0: Clears the tilted working plane indexing.
(2) Bit 6 (CLR) of parameter No. 3402 = 1: Clears with a reset.
(3) Bit 0 (C08) of parameter No. 3407 = 0: Clears the G code in group 08.
(4) Bit 7 (C23) of parameter No. 3408 = 0: Clears the G code in group 23.
(5) Bit 7 (CFH) of parameter No. 3409 = 0: Clears the F, H, D, and T codes.
(6) Bit 6 (LVK) of parameter No. 5003 = 0: Clears the tool length compensation vector.

Look-ahead acceleration/deceleration before interpolation

When using tool center point retention type tool axis direction control, use acceleration/deceleration before look ahead interpolation at the same time. If acceleration/deceleration before look ahead interpolation is not used, a shock may result. Furthermore, be sure to set the following parameters:

(1) Bit 1 (LRP) of parameter No. 1401 = 1:
Linear rapid traverse

(2) Bit 5 (FRP) of parameter No. 19501 = 1:
For rapid traverse, uses look-ahead acceleration/deceleration before interpolation.
(3) Parameter No. 1671:
Acceleration of acceleration/deceleration for rapid traverse.
(4) Parameter No. 1672:
Acceleration change time of look-ahead bell-shaped acceleration/deceleration before interpolation for rapid traverse.
(5) Parameter No. 1660:
Maximum permissible acceleration of look-ahead acceleration/deceleration before interpolation.

If any of these settings is not made, alarm PS5458, "ILLEGAL USE OF G53.1/G53.6", occurs.

- Relationships with other modal commands

G41, G42, and G40 (cutter compensation), G43, G49 (tool length compensation), G51.1 and G50.1 (programmable mirror image), and canned cycle commands must have nesting relationships with G68.2.
In other words, first issue G68.2 when the modes mentioned above are off, turn the modes on and off, then issue G69.

- Manual reference position return

Execution of manual reference position return in the tilted working plane indexing mode issues alarm PS5324, “REFERENCE RETURN INCOMPLETE”. Before performing manual reference position return, cancel the tilted working plane indexing mode.

- Hypothetical axis of a table rotation axis

When a table rotation axis is set as a hypothetical axis, tilted working plane indexing is performed on the assumption that the table rotation axis is at 0 degrees.

- Specifiable G codes

The G codes that can be specified in the tilted working plane indexing mode are listed below.
Specifying a G code other than these codes results in alarm PS5462 “ILLEGAL COMMAND (G68.2/G69)”.

- Positioning (G00)
- Linear interpolation (G01)
- Circular interpolation / helical interpolation (G02/G03)
- Dwell (G04)
- AI contour control(G05.1 Q0/Q1)
- Exact stop (G09)
- Programmable data input (G10)
- Tool retract and recover (G10.6)
- Programmable data input mode cancel (G11)
- Plane selection (G17/G18/G19)
- Automatic return to reference position (G28)
- Movement from reference position (G29)
- 2nd, 3rd and 4th reference position return (G30)
- Skip function (G31)
- Cutter compensation : cancel (G40)
- Tool radius or tool nose radius compensation (G41/G42)
- Tool length compensation + (G43)
- Tool length compensation - (G44)
- Tool length compensation cancel (G49,G49.1)
- Programmable mirror image cancel (G50.1)
- Local coordinate system (G52)
- Machine coordinate system setting (G53)
- Tool axis direction control (G53.1)
- Workpiece coordinate system selection (G54 to G59, G54.1)

- Exact stop mode (G61)
- Tapping mode (G63)
- Cutting mode(G64)
- Macro call (G65)
- Macro modal call A (G66)
- Macro modal call B (G66.1)
- Macro modal call A/B cancel (G67)
- Canned cycle for drilling (G73, G74, G76, G80 to G89)
- Absolute programming (G90)
- Incremental programming (G91)
- Constant surface speed control (G96)
- Constant surface speed control cancel (G97)
- Canned cycle : return to initial level (G98)
- Canned cycle : return to R point level (G99)

M

- Tool offset increase (G45)
- Tool offset decrease (G46)
- Tool offset double increase (G47)
- Tool offset double decrease (G48)
- Scaling cancel (G50)
- Scaling (G51)
- Programmable mirror image (G51.1)
- Automatic corner override (G62)
- Mode off of coordinate system rotation or 3-dimensional coordinate system conversion (G69)
- Inverse time feed (G93)
- Feed per minute (G94)
- Feed per revolution (G95)

T

- Mode off of coordinate system rotation or 3-dimensional coordinate system conversion (G69.1)
- Feed per minute (G98 (G94))
- Feed per revolution (G99 (G95))

- Modal G codes that allow specification of a tilted working plane indexing

A tilted working plane indexing can be specified in the modal G code states listed below.
In a modal state other than the following modal G codes, specifying the tilted working plane indexing results in alarm PS5462 “ILLEGAL COMMAND (G68.2/G69)”:

- Positioning (G00)
- Linear interpolation (G01)
- Programmable data input mode cancel (G11)
- Plane selection (G17/G18/G19)
- Polar coordinates command cancel (G15)
- Input in inch (G20 (G70))
- Input in mm (G21 (G71))
- Stored stroke check function (G22/G23)
- Cutter compensation : cancel (G40)
- Tool length compensation cancel (G49,G49.1)
- Programmable mirror image cancel (G50.1)
- Workpiece coordinate system selection (G54 to G59, G54.1)
- Exact stop mode (G61)
- Tapping mode (G63)
- Cutting mode (G64)
- Macro modal call A/B cancel (G67)

- Canned cycle cancel (G80)
- Absolute programming (G90)
- Incremental programming (G91)
- Constant surface speed control cancel (G97)
- Canned cycle : return to initial level (G98)
- Canned cycle : return to R point level (G99)

M

- Scaling cancel (G50)
- Automatic corner override (G62)
- Mode off of coordinate system rotation or 3-dimensional coordinate system conversion (G69)
- Inverse time feed (G93)
- Feed per minute (G94)
- Feed per revolution (G95)

T

- Polar coordinate interpolation mode cancel (G13.1)
- Polygon turning cancel (G50.2)
- Mirror image for double turret off/balanced cutting mode cancel (G69)
- Mode off of coordinate system rotation or 3-dimensional coordinate system conversion (G69.1)
- Feed per minute (G98 (G94))
- Feed per revolution (G99 (G95))
- Polar coordinate interpolation mode cancel (G113)

Alarm and message

Number	Message	Description
PS0438	ILLEGAL PARAMETER IN TOOL DIRC CMP	If on a 5-axis machine, a case below applies, a parameter is illegal. Tool center point retention type tool axis direction control (G53.6) is performed. Parameter settings - Look-ahead acceleration/deceleration before interpolation is disabled. Set parameter No. 1660. - Rapid traverse acceleration/deceleration before interpolation is disabled. Set bit 1 of parameter No. 1401, bit 5 of parameter No. 19501, and parameter No. 1671. - Any of machine configuration parameters Nos. 19680 to 19714 is illegal. - The axis set in parameter No. 19681 or 19686 is not a rotation axis. - The basic three axes are not set in parameter No. 1022.
PS5324	REFERENCE RETURN INCOMPLETE	Manual reference position return cannot be performed during three-dimensional coordinate conversion, execution of the tilted working plane indexing.
PS5456	TOO MANY G68.2 NESTING	Tilted working plane indexing G68.2 was specified more than once. To perform another coordinate conversion, perform cancellation, then specify the coordinate conversion.
PS5457	G68.2/G68.3 FORMAT ERROR	A G68.2/G68.3 format error occurred.
PS5458	ILLEGAL USE OF G53.1/G53.6	- G53.1/G53.6 is specified before a G68.2 command. - G53.1/G53.6 is not specified alone. - No tool is specified with a G53.6 command.

Number	Message	Description
PS5459	MACHINE PARAMETER INCORRECT	- A machine configuration parameter (parameters Nos. 19665 to 19667, 19680 to 19714, or 12321) is incorrect. - The axis set in parameter No. 19681 or 19686 is not a rotation axis. - The basic three axes are not set in parameter No. 1022. - A rotary axis end point found by the NC with the tilted working plane indexing is not within the range set by parameters No 19741 to 19744. - When tool axis direction control (G53.1/G53.6) is specified on the following condition that parameter is set incorrectly if bit 7 (AIR) of parameter No.11221 is set 1. - In case that the movement range of rotation axis is set to the roll-over axis - In case that roll-over function is set to rotary axis (B type)
5462	ILLEGAL COMMAND (G68.2/G69)	(1) The modal setting used when G68.2 or G69 is specified is incorrect. (2) An unspecifiable G code was specified in the G68.2 mode. (3) The offset vector of tool radius/tool nose radius compensation is not canceled when G68.2 or G69 is specified.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Tilted working plane indexing

12.17 MACRO COMPILER/MACRO EXECUTER

Overview

The macro executor function converts custom macros created by machine tool builders to executable programs, registers them in the FLASH ROM module, and executes them to solve problems as described below.
NC programs are divided into two types: Programs that are hardly modified after created (programs created using custom macros) and programs that differ depending on the workpiece (machining programs). Since different types of programs are processed in the same way, a custom macro may be destroyed due to a battery failure or operator error.

Features

- Since the program is stored in executable form program, the execution speed is high. Machine time is then reduced, and precision is improved.
- Since the program is stored in FLASH ROM, there is no lost of data of battery failure or corruption. Reliability is improved.
- Stored programs are not displayed on the program screen, so know-how of the machine tool builders is kept protected.
- Since custom macros are stored in FLASH ROM, the program editing memory can be used effectively.
- The user can call the macro without knowing the stored program. A custom macro can be created and executed in the program edit memory.
- An original screen can be created by using the graphic display or by selecting screens by the soft key. The machine tool builder can extend the control function by using such functions as machine program creation and edit control, RS-232C interface control, and PMC data read/write functions.

Reference item

Macro Executor PROGRAMMING MANUAL (B-63943EN-2)

12.18 OPTIONAL ANGLE CHAMFERING AND CORNER ROUNDING

M

Overview

Chamfering and corner rounding blocks can be inserted automatically between the following:

- Between linear interpolation and linear interpolation blocks
- Between linear interpolation and circular interpolation blocks
- Between circular interpolation and linear interpolation blocks
- Between circular interpolation and circular interpolation blocks

It is possible to use this function with AI contour control.

Limitation

- Macro executer

Optional angle chamfering and corner rounding specified in the execution macro is disabled.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5105								SBC

[Input type] Parameter input
[Data type] Bit path

#0 SBC In a drilling canned cycle, chamfer cycle, or corner rounding cycle:
0: A single block stop is not performed.
1: A single block stop is performed.

Alarm and message

Number	Message	Description
PS0050	CHF/CNR NOT ALLOWED IN THRD BLK	Chamfering or corner R is commanded in the thread cutting block. Modify the program.
PS0051	MISSING MOVE AFTER CNR/CHF	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.
PS0055	MISSING MOVE VALUE IN CHF/CNR	In chamfering or corner R block, the move distance is less than chamfer or corner R amount. Modify the program.

12.19 CHAMFERING AND CORNER ROUNDING

T

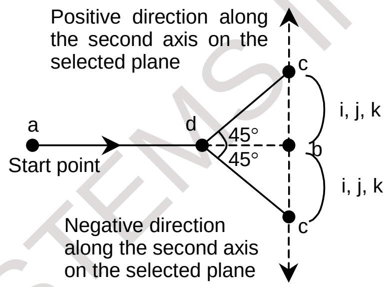
Overview

A chamfering or corner rounding block can automatically be inserted between linear interpolation (G01) along a single axis and that along a single axis normal to that single axis.
Chamfering or corner rounding is inserted for a command to move the tool along two axes on the plane determined by the plane selection (G17, G18, or G19) command.

Format

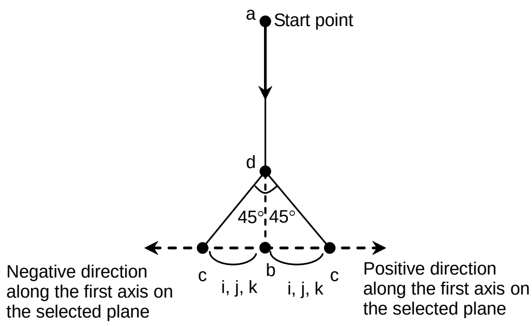
- Chamfering
First axis on the selected plane → second axis on the selected plane
(G17 plane: $X_P \rightarrow Y_P$, G18 plane: $Z_P \rightarrow X_P$, G19 plane: $Y_P \rightarrow Z_P$)

Format	
G17 plane:	**G01 X_P(U)_ J(C)±j ;**
G18 plane:	**G01 Z_P(W)_ I(C)±i ;**
G19 plane:	**G01 Y_P(V)_ K(C)±k ;**

Explanation		**Tool movement**
X_P(U)__ **Y_P(V)__** **Z_P(W)__**	Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right. X_P is the address of the X-axis of the three basic axes or an axis parallel to the X-axis. Y_P is the address of the Y-axis of the three basic axes or an axis parallel to the Y-axis. Z_P is the address of the Z-axis of the three basic axes or an axis parallel to the Z-axis.	
I(C)±i **J(C)±j** **K(C)±k**	Specify the distance between points b and c in the figure shown at right with a sign following address I, J, K, or C. (Use I, J, or K when bit 4 (CCR) of parameter No. 3405 is set to 0 or C when the bit is set to 1.)	Moves from a to d and c. (Positive direction along the second axis on the selected plane when a plus sign is specified at I, J, K, or C or negative direction when a minus sign is specified at I, J, K, or C)

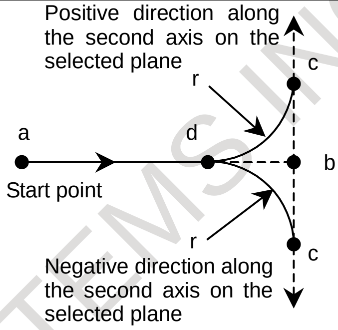
- Chamfering
Second axis on the selected plane → first axis on the selected plane
(G17 plane: $Y_P \rightarrow X_P$, G18 plane: $X_P \rightarrow Z_P$, G19 plane: $Z_P \rightarrow Y_P$)

Format	
G17 plane:	**G01 Y_P(V)_ I(C)±i ;**
G18 plane:	**G01 X_P(U)_ K(C)±k ;**
G19 plane:	**G01 Z_P(W)_ J(C)±j ;**

Explanation		**Tool movement**
X_P(U)__ **Y_P(V)__** **Z_P(W)__**	Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right. X_P is the address of the X-axis of the three basic axes or an axis parallel to the X-axis. Y_P is the address of the Y-axis of the three basic axes or an axis parallel to the Y-axis. Z_P is the address of the Z-axis of the three basic axes or an axis parallel to the Z-axis.	Moves from a to d and c. (Positive direction along the first axis on the selected plane when a plus sign is specified at I, J, K, or C or negative direction when a minus sign is specified at I, J, K, or C)
I(C)±i **J(C)±j** **K(C)±k**	Specify the distance between points b and c in the figure shown at right with a sign following address I, J, K, or C. (Use I, J, or K when bit 4 (CCR) of parameter No. 3405 is set to 0 or C when the bit is set to 1.)	

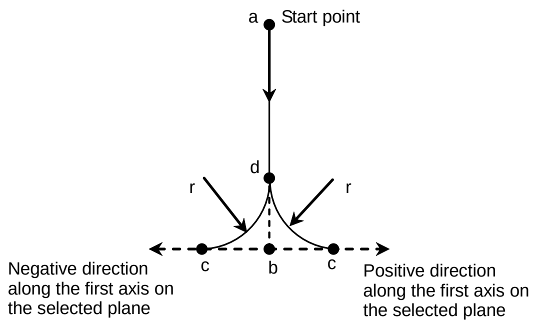
- Corner R
First axis on the selected plane → second axis on the selected plane
(G17 plane: $X_P \rightarrow Y_P$, G18 plane: $Z_P \rightarrow X_P$, G19 plane: $Y_P \rightarrow Z_P$)

Format
G17 plane: G01 X_P(U)_ R±r ; **G18 plane: G01 Z_P(W)_ R±r ;** **G19 plane: G01 Y_P(V)_ R±r ;**

Explanation		Tool movement
X_P(U)__ **Y_P(V)__** **Z_P(W)__**	Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right. X_P is the address of the X-axis of the three basic axes or an axis parallel to the X-axis. Y_P is the address of the Y-axis of the three basic axes or an axis parallel to the Y-axis. Z_P is the address of the Z-axis of the three basic axes or an axis parallel to the Z-axis.	
R±r	Specify the radius of the arc connecting points d and c in the figure shown at right with a sign following address R.	Moves from a to d and c. (Positive direction along the second axis on the selected plane when +r is specified at R or negative direction when -r is specified at R)

- Corner R
Second axis on the selected plane → first axis on the selected plane
(G17 plane: $Y_P \rightarrow X_P$, G18 plane: $X_P \rightarrow Z_P$, G19 plane: $Z_P \rightarrow Y_P$)

Format
G17 plane: G01 Y_P(V)_ R±r ; **G18 plane: G01 X_P(U)_ R±r ;** **G19 plane: G01 Z_P(W)_ R±r ;**

Explanation		Tool movement
X_P(U)__ **Y_P(V)__** **Z_P(W)__**	Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right. X_P is the address of the X-axis of the three basic axes or an axis parallel to the X-axis. Y_P is the address of the Y-axis of the three basic axes or an axis parallel to the Y-axis. Z_P is the address of the Z-axis of the three basic axes or an axis parallel to the Z-axis.	Moves from a to d and c. (Positive direction along the first axis on the selected plane when +r is specified at R or negative direction when -r is specified at R)
R±r	Specify the radius of the arc connecting points d and c in the figure shown at right with a sign following address R.	

Explanation

By G01 specified for chamfering or corner R, the tool must be moved only along one of the two axes on the selected plane. The command in the next block must move the tool only along the other axis on the selected plane.

Example:

When the A-axis is set as an axis parallel to the basic X-axis (by setting parameter No. 1022 to 5), the following program performs chamfering between cutting feed along the A-axis and that along the Z-axis:

```
G18 A0 Z0
G00 A100.0 Z100.0
G01 A200.0 F100 K30.0
Z200.0
```

The following program causes an alarm. (Because chamfering is specified in the block to move the tool along the X-axis, which is not on the selected plane)

```
G18 A0 Z0
G00 A100.0 Z100.0
G01 X200.0 F100 K30.0
Z200.0
```

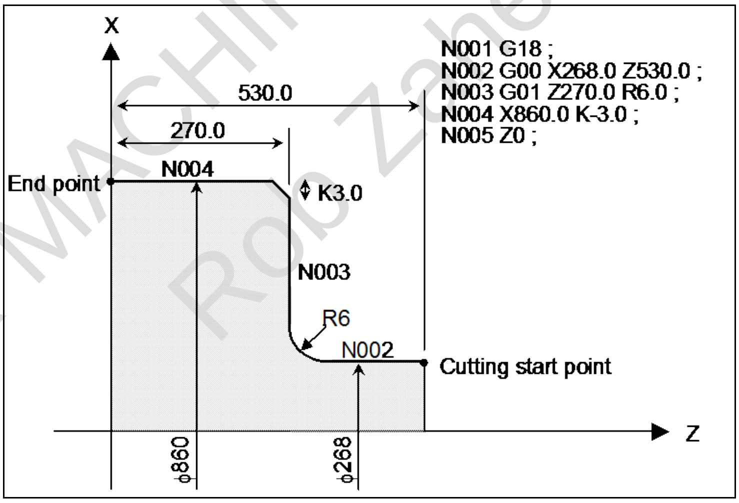
The following program also causes an alarm. (Because the block next to the chamfering command moves the tool along the X-axis, which is not on the selected plane)

```
G18 A0 Z0
G00 A100.0 Z100.0
G01 Z200.0 F100 I30.0
X200.0
```

A radius value is specified at I, J, K, R, and C.

In an incremental programming, use point b in the figure in "Format" as the start point in the block next to a chamfering or corner R block. That is, specify the distance from point b. Do not specify the distance from point c.

Example



Limitation

- Alarms

In the following cases, an alarm is issued:

1) Chamfering or corner R is specified in a block for threading (alarm PS0050, “CHF/CNR NOT ALLOWED IN THRD BLK”).

2) G01 is not specified in the block next to the G01 block in which chamfering or corner R is specified (alarm PS0051, “MISSING MOVE AFTER CNR/CHF” or PS0052, “CODE IS NOT G01 AFTER CHF/CNR”).

3) An axis which is not on the selected plane is specified as a move axis in the block in which chamfering or corner R is specified or the next block (alarm PS0051 or PS0052).

4) A plane selection command (G17, G18, or G19) is specified in the block next to the block in which chamfering or corner R is specified (alarm PS0051).

5) When bit 4 (CCR) of parameter No. 3405 is set to 0 (to specify chamfering at I, J, or K), two or more of I, J, K, and R are specified in G01 (alarm PS0053, "TOO MANY ADDRESS COMMANDS").

6) Chamfering or corner R is specified in the G01 block to move the tool along more than one axis (alarm PS0054, "NO TAPER ALLOWED AFTER CHF/CNR").

7) The travel distance along an axis specified in the block in which chamfering or corner R is specified is smaller than the amount of chamfering or corner R (alarm PS0055, "MISSING MOVE VALUE IN CHF/CNR"). (See the Fig. 12.19 (a).)

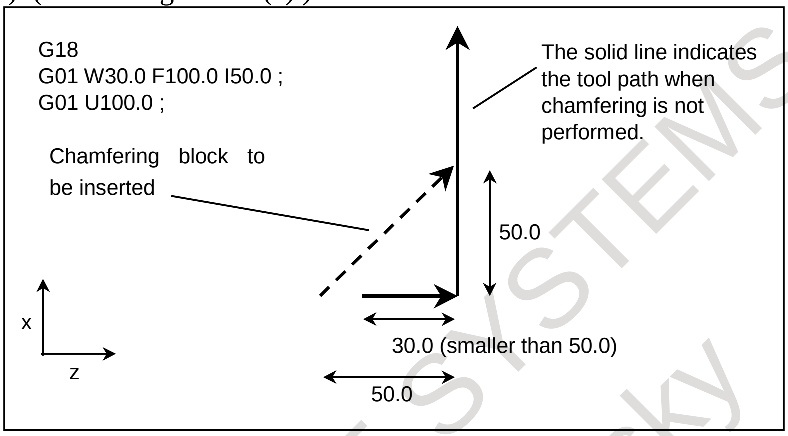


Fig. 12.19 (a) Example of machining which causes alarm PS0055

8) An invalid combination of a move axis and I, J, or K is specified for chamfering (alarm PS0306, "MISMATCH AXIS WITH CNR/CHF").

9) An invalid sign is specified at I, J, K, R, or C (chamfering or corner R in the direction opposite to the movement in the next block is specified) (alarm PS0051). (See the Fig. 12.19 (b).)

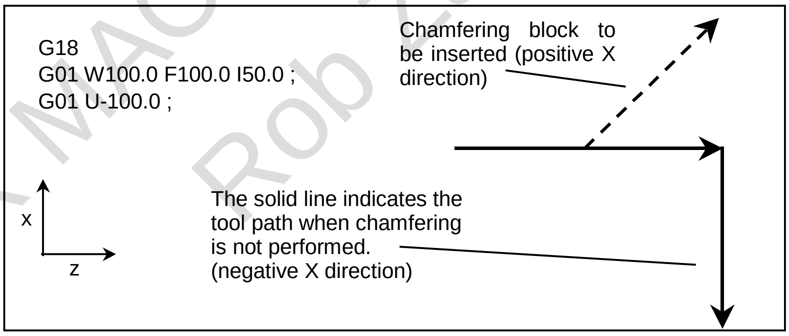


Fig. 12.19 (b) Example of machining which causes alarm PS0051

10) In a shape program in the multiple repetitive canned cycle (G70, G71, G72, or G73), a command for the chamfering or corner R in the last block is terminated in the middle (alarm PS0069, "LAST BLOCK OF SHAPE PROGRAM IS AN ILLEGAL COMMAND").

- Single block operation

When the block in which chamfering or corner R is specified is executed in the single block mode, operation continues to the end point of the inserted chamfering or corner R block and the machine stops in the feed hold mode at the end point. When bit 0 (SBC) of parameter No. 5105 is set to 1, the machine stops in the feed hold mode also at the start point of the inserted chamfering or corner R block.

- Macro executer

Chamfering and corner rounding specified in the execution macro is disabled.

- Cutter compensation or tool nose radius compensation

When applying cutter or tool nose radius compensation, note the following points:

1. If the amount of inner chamfering or corner R is too small as compared with compensation and cutting is generated, alarm PS0041, “INTERFERENCE IN CUTTER COMPENSATION” is issued. (See the Fig. 12.19 (c).)

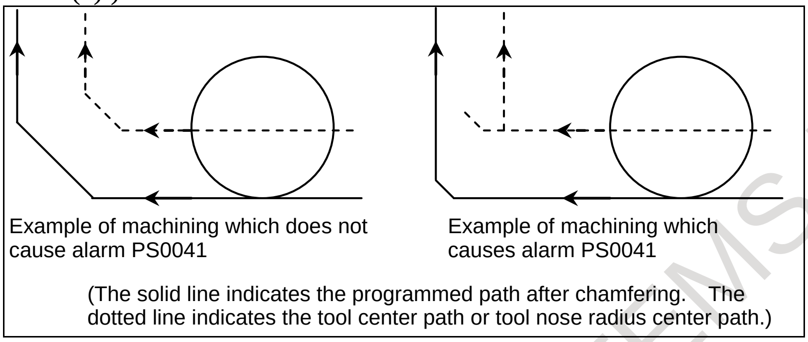


Fig. 12.19 (c)

2. A function is available which intentionally changes the compensation direction by specifying the I, J, or K command in the G01 block in the cutter or tool nose radius compensation mode (see the explanations of cutter or tool nose radius compensation). To use this function when the chamfering and corner R function is used, set bit 4 (CCR) of parameter No. 3405 to 1 so that I, J, and K are not used for specifying chamfering. Operation to be performed under each condition is explained below.

   (1) When the chamfering and corner R function is not used
   In the G01 block in the cutter or tool nose radius compensation mode, the cutter or tool nose radius compensation direction can be specified at address I, J, or K.
   No chamfering is performed.

   (2) When the chamfering and corner R function is used
   (2-1) When bit 4 (CCR) of parameter No. 3405 is set to 0
   In the G01 block in the cutter or tool nose radius compensation mode, chamfering can be specified at address I, J, or K. Corner R can also be specified at address R.
   The cutter or tool nose radius compensation direction cannot be specified.
   (2-2) When bit 4 (CCR) of parameter No. 3405 is set to 1
   In the G01 block in the cutter or tool nose radius compensation mode, the cutter or tool nose radius compensation direction can be specified at address I, J, or K.
   Chamfering or corner R can also be specified at address C or R.

> **NOTE**
> The chamfering and corner R function and direct drawing dimension programming function cannot be used simultaneously. When bit 0 (CRD) of parameter No. 3453 is set to 0, chamfering and corner R is enabled. (At this time, direct drawing dimension programming is disabled.)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3405				CCR				

[Input type] Parameter input
[Data type] Bit path

#4 CCR Addresses used for chamfering

0: Address is “I”, “J”, or “K”.
In direct drawing dimension programming, addresses ",C", ",R", and ",A" (with comma) are used in stead of "C", "R", and "A".

1: Address is “C”.
Addresses used for direct drawing dimension programming are "C", "R", and "A" without comma.

> **NOTE**
> If this bit (CCR) is set to 0, the function for changing the compensation direction by specifying I, J, or K in a G01 block in the cutter compensation/ tool nose radius compensation mode cannot be used.
> If this bit (CCR) is set to 1 when address C is used as an axis name, the chamfer function cannot be used.

	#7	#6	#5	#4	#3	#2	#1	#0
3453								CRD

[Input type] Setting input
[Data type] Bit path

#0 CRD About the functions of chamfering or corner R and direct drawing dimension programming,

0: Chamfering or corner R is enabled.
1: Direct drawing dimension programming is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5105								SBC

[Input type] Parameter input
[Data type] Bit path

#0 SBC In a drilling canned cycle, chamfer cycle, or corner rounding cycle:

0: A single block stop is not performed.
1: A single block stop is performed.

Alarm and message

Number	Message	Description
PS0050	CHF/CNR NOT ALLOWED IN THRD BLK	Chamfering or corner R is commanded in the thread cutting block. Modify the program.
PS0051	MISSING MOVE AFTER CNR/CHF	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.
PS0052	CODE IS NOT G01 AFTER CHF/CNR	The block next to the chamfering or corner R block is not G01 (or vertical line). Modify the program.
PS0053	TOO MANY ADDRESS COMMANDS	In the chamfering and corner R commands, two or more of I, J, K and R are specified.
PS0054	NO TAPER ALLOWED AFTER CHF/CNR	A block in which chamfering or the corner R was specified includes a taper command. Modify the program.
PS0055	MISSING MOVE VALUE IN CHF/CNR	In chamfering or corner R block, the move distance is less than chamfer or corner R amount. Modify the program.

Number	Message	Description
PS0069	LAST BLOCK OF SHAPE PROGRAM IS AN ILLEGAL COMMAND	In a shape program in the multiple repetitive canned cycle (G70, G71, G72, or G73), a command for the chamfering or corner R in the last block is terminated in the middle.
PS0306	MISMATCH AXIS WITH CNR/CHF	The correspondence between the moving axis and the I, J, or K command is incorrect in a block in which chamfering is specified.

12.20 DIRECT DRAWING DIMENSIONS PROGRAMMING

T

Overview

Angles of straight lines, chamfering value, corner rounding values, and other dimensional values on machining drawings can be programmed by directly inputting these values. In addition, the chamfering and corner rounding can be inserted between straight lines having an optional angle.
This programming is only valid in memory operation mode.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3405			DDP	CCR				

[Input type] Parameter input
[Data type] Bit path

#4 CCR Addresses used for chamfering

0: Address is "I", "J", or "K".
In direct drawing dimension programming, addresses ",C", ",R", and ",A" (with comma) are used in stead of "C", "R", and "A".

1: Address is "C".
Addresses used for direct drawing dimension programming are "C", "R", and "A" without comma.

> **NOTE**
> If this bit (CCR) is set to 0, the function for changing the compensation direction by specifying I, J, or K in a G01 block in the cutter compensation/ tool nose radius compensation mode cannot be used.
> If this bit (CCR) is set to 1 when address C is used as an axis name, the chamfer function cannot be used.

#5 DDP Angle commands by direct drawing dimension programming

0: Normal specification
1: A supplementary angle is given.

	#7	#6	#5	#4	#3	#2	#1	#0
3453								CRD

[Input type] Setting input
[Data type] Bit path

#0 CRD About the functions of chamfering or corner R and direct drawing dimension programming,
0: Chamfering or corner R is enabled.
1: Direct drawing dimension programming is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3458	CRC							

[Input type] Parameter input
[Data type] Bit path

#7 CRC In Direct drawing dimension programming function, when a minus value is commanded as a chamfering value / corner R value:
0: Alarm PS0055 “MISSING MOVE VALUE IN CHF/CNR” is issued.
1: Alarm is not issued. A minus value is converted to a plus value.

Alarm and message

Number	Message	Description
PS0055	MISSING MOVE VALUE IN CHF/CNR	In chamfering or corner R block, the move distance is less than chamfer or corner R amount. Modify the program.
PS0056	NO END POINT & ANGLE IN CHF/CNR	In direct dimension drawing programming, both an end point and an angle were not specified in the block next to the block in which only an angle was specified (Aa). Modify the program.
PS0057	NO SOLUTION OF BLOCK END	Block end point is not calculated correctly in direct dimension drawing programming. Modify the program.
PS0058	END POINT NOT FOUND	Block end point is not found in direct dimension drawing programming. Modify the program.
PS0312	ILLEGAL COMMAND IN DIRECT DRAWING DIMENSIONS PROGRAMMING	Direct input of drawing dimensions was commanded in an invalid format. An attempt was made to specify an invalid G code during direct input of drawing dimensions. There exist two or more blocks not to be moved in consecutive commands that specify direct input of drawing dimensions. Although non-use of commas (,) (bit 4 of parameter No. 3405 = 1) was specified for direct input of drawing dimensions, a comma was specified.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Direct drawing dimensions programming

12.21 PATTERN DATA INPUT

Overview

In the program of the fixed form processing with the custom macro, the operator select the processing pattern on the menu screen and specified the size, number and so on to the variable on the custom macro screen. As above mentioned, this function enables users to perform programming simply without programming using an existing NC language.
With the aid of this function, a machine tool builder can prepare the program of a hole machining cycle (such as a boring cycle or tapping cycle) using the custom macro function, and can store it into the program memory.
This cycle is assigned pattern names, such as BOR1, TAP3, and DRL2.
An operator can select a pattern from the menu of pattern names displayed on the screen.
Data (pattern data) which is to be specified by the operator should be created in advance with variables in a drilling cycle.
The operator can identify these variables using names such as DEPTH, RETURN RELIEF, FEED, MATERIAL or other pattern data names. The operator assigns values (pattern data) to these names.
The operator selects the pattern on the menu screen, and the selected pattern number is assigned to the system variable. The custom macro of the selected pattern can be started by starting a program then referring to the system variable in the program.

Explanation

This function consists of Pattern menu screen and Custom macro screen.
The process pattern is selected on the pattern screen.
Then the process pattern is selected, the custom macro screen is displayed.
On this custom macro screen, the variable with the name and comment is displayed according to the selected process pattern.
The process data can be input by referring to the variable name with the numerical value on the drawing.

A sample pattern menu screen and custom macro screens are shown below.

(1) Pattern menu screen

Fig. 12.21 (a) Pattern menu screen

(2) Custom macro screen

The name of variable and comment can be displayed on the usual custom macro screen.

The menu title and pattern name on the pattern menu screen and the variable name on the custom macro screen can be defined

The position of the comment displayed on the custom macro screen can be selected whether the lower size or the right side by setting bit 0 (POC) of parameter No. 11318.

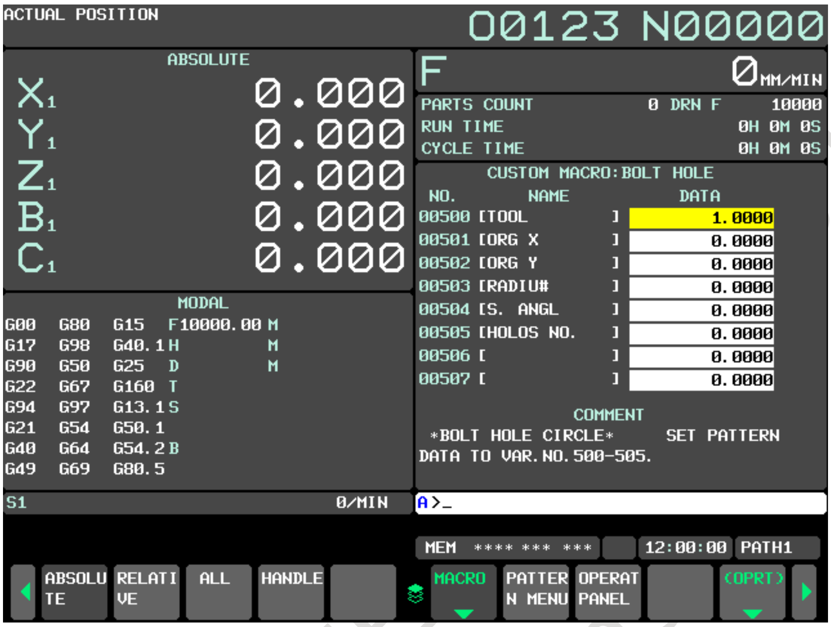


Fig. 12.21 (b) Custom macro screen (parameter No. 11318#0=0)

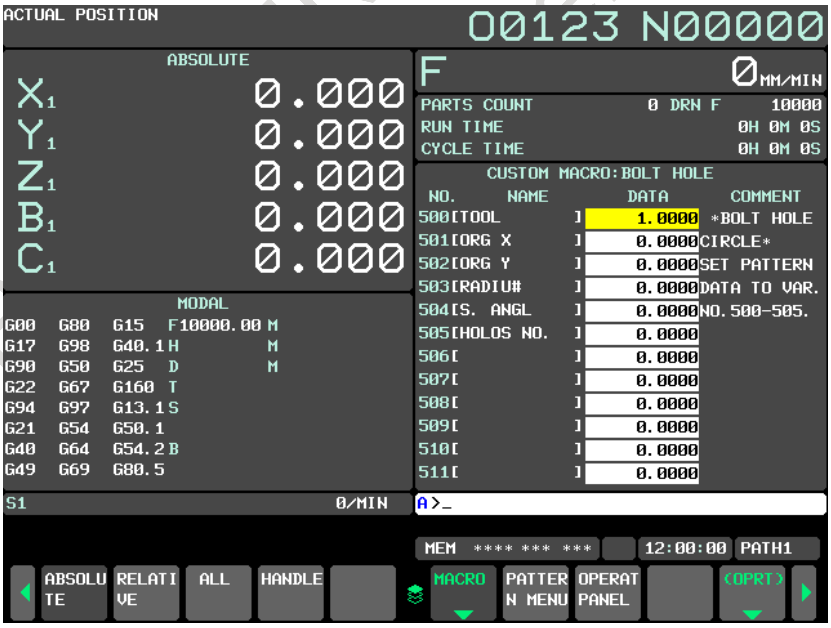


Fig. 12.21 (c) Custom macro screen (parameter No. 11318#0=1)

Explanation of operation

The following steps 1-3 explain how to display the pattern menu screen.

1 Press function key .

2 Press continuous menu key .

3 Press soft key [PATTERN MENU].

- Pattern menu screen

The pattern menu in the Fig. 12.21 (d) is displayed.

Fig. 12.21 (d) Pattern menu screen

Select the pattern on this screen
The next two methods are effective.

- Selection by cursor

  Move the cursor to the pattern name with the cursor move keys , and press the soft key [SELECT] or key.

- Selection by setting of pattern number

  Enter the number on the left of the pattern name, and press the soft key [SELECT] or key.

The selected pattern number is registered to system variable #5900. The custom macro of the selected pattern can be started by starting a fixed program (external program No. search) with an external signal. This program refers to the system variable #5900 in the program. This system variable #5900 is kept after power-off.

- Custom macro variable screen

The custom macro screen in the Fig. 12.21 (e) is displayed.

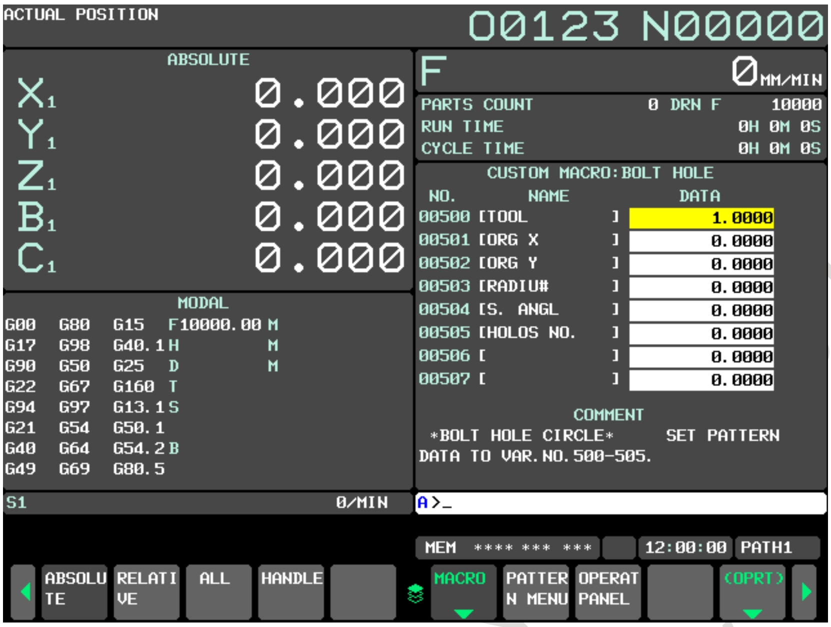


Fig. 12.21 (e) Custom macro screen when the pattern data is input

When the screen is changed to the custom macro screen, the macro variable number that is selected first is specified with the parameters Nos.6101 to 6110. The macro variables that variable name is not defined can be input, too.

> **NOTE**
> 1 The variable name that is displayed cannot be used as the common variable name of the NC program.
> 2 When the common variable name is defined by SETVN command, the variable name defined by pattern data input function is given priority.
> 3 When the bit 0 (POC) of parameter No. 11318 is set to 1, The variable number is three digit display.
> And the value of 12 digits or more is input, 11 digits from head of value are displayed.
> Example)
> Input: -123456789.123 → Display: -123456789.1

Parameter

6101	Macro variable number selected first when pattern menu 1 is selected
6102	Macro variable number selected first when pattern menu 2 is selected
6103	Macro variable number selected first when pattern menu 3 is selected
6104	Macro variable number selected first when pattern menu 4 is selected
6105	Macro variable number selected first when pattern menu 5 is selected
6106	Macro variable number selected first when pattern menu 6 is selected
6107	Macro variable number selected first when pattern menu 7 is selected
6108	Macro variable number selected first when pattern menu 8 is selected

6109	Macro variable number selected first when pattern menu 9 is selected
6110	Macro variable number selected first when pattern menu 10 is selected

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0,100 to 199,500 to 999

Set the macro variable number to be selected first when a pattern menu is selected on the custom macro screen.
If 0 is specified, 500 is assumed.
If a value beyond the above range is entered, 100 is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
11318								POC

[Input type] Parameter input
[Data type] Bit

#0 POC When the pattern data input function is used, on the custom macro screen a comment is:
0: Displayed in the lower part of the screen.
1: Displayed on the right side of the screen.

> **NOTE**
> This parameter is not used when a 15-/19-inch display unit is used.

Definition of the screen

The definition of the screen is performed by NC program.

- Program configuration

This function is consist of one program for the definition of pattern menu screen and maximum ten programs for the definition of custom macro screen.
Register the macro program for the pattern data input function to the following folder.
//CNC_MEM/USER/LIBRARY/PATH1 to PATH[n]
* [n] is the maximum path number of the system
* The PATH1 to PATH[n] should be made by the machine tool builder.

The program number is as follows

Table 12.21 (a) Program Number and screen

Program No.	Screen
O9500	Program to specify character strings displayed on the pattern menu screen.
O9501	Program to specify character string displayed on the custom macro screen for pattern No.1
O9502	Program to specify character string displayed on the custom macro screen for pattern No.2
O9503	Program to specify character string displayed on the custom macro screen for pattern No.3
O9504	Program to specify character string displayed on the custom macro screen for pattern No.4
O9505	Program to specify character string displayed on the custom macro screen for pattern No.5
O9506	Program to specify character string displayed on the custom macro screen for pattern No.6
O9507	Program to specify character string displayed on the custom macro screen for pattern No.7
O9508	Program to specify character string displayed on the custom macro screen for pattern No.8

Program No.	Screen
O9509	Program to secify character string displayed on the custom macro screen for pattern No.9
O9510	Program to secify character string displayed on the custom macro screen for pattern No.10

Table 12.21 (b) Macro commands used in the pattern data input function

G code	H code	Function
G65	H90	Specifies the menu title.
G65	H91	Specifies the pattern name.
G65	H92	Specifies the pattern data title.
G65	H93	Specifies the variable name.
G65	H94	Specifies the comment.

Table 12.21 (c) System variables used in the pattern data input function

System variable	Description
#5900	Pattern No. selected by user.

> **NOTE**
> The pattern menu screen and the custom macro screen are defined only by preserving programs O9500 to O9510 in the above-mentioned folder, and these programs need not be executed. If these programs are executed, alarm (PS0074), "ILLEGAL PROGRAM NUMBER" is issued.

Definition of the pattern menu screen

Menu title and pattern name are defined as follows.

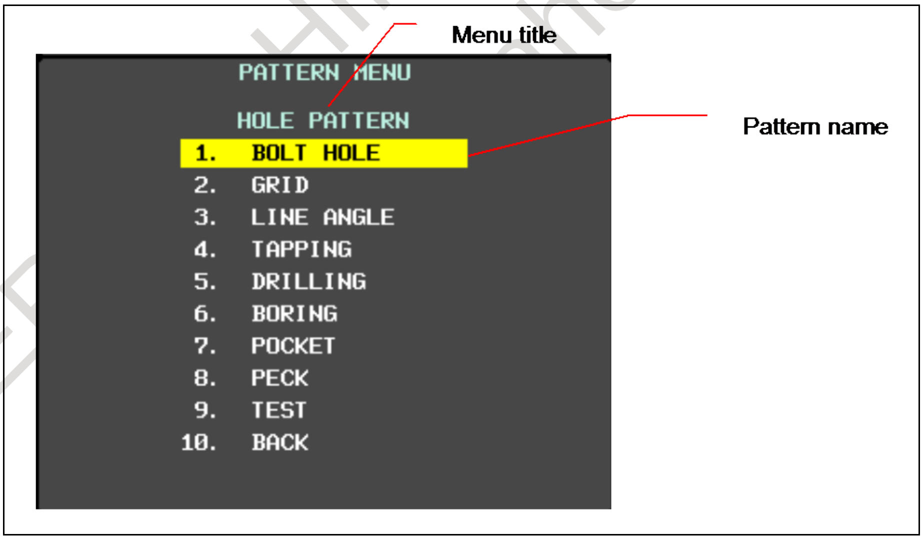


Fig. 12.21 (f) Pattern menu screen

Definition of menu title

The character string displayed in the menu title of the pattern menu screen is defined.

The menu title is specified up to 12 characters in a half size letter and up to 6 characters in a full size letter.

- Format

G65 H90 P_ Q_ R_ I_ J_ K_ ;

H90	:	Specifies the menu title
P_	:	The code of 1st and 2nd characters of title
Q_	:	The code of 3rd and 4th characters of title
R_	:	The code of 5th and 6th characters of title
I_	:	The code of 7th and 8th characters of title
J_	:	The code of 9th and 10th characters of title
K_	:	The code of 11th and 12th characters of title

As for the way of setting the character-code, refer to the Subsection, "Setting the Character-codes".

Definition of pattern name

The character string displayed in the pattern name which becomes a menu item is defined.
The pattern name is specified up to 10 characters in a half size letter and up to 5 characters in a full size letter.

- Format

G65 H91 P_ Q_ R_ I_ J_ K_ ;

H91	:	Specifies the pattern name
P_	:	Specifies the menu number of the pattern name The menu number = 1 to 10
Q_	:	The code of 1st and 2nd characters of pattern name
R_	:	The code of 3rd and 4th characters of pattern name
I_	:	The code of 5th and 6th characters of pattern name
J_	:	The code of 7th and 8th characters of pattern name
K_	:	The code of 9thd and 10th characters of pattern name

As for the way of setting the character-code, refer to the Subsection, "Setting the Character-codes".

Example

The following is example for pattern menu screen.

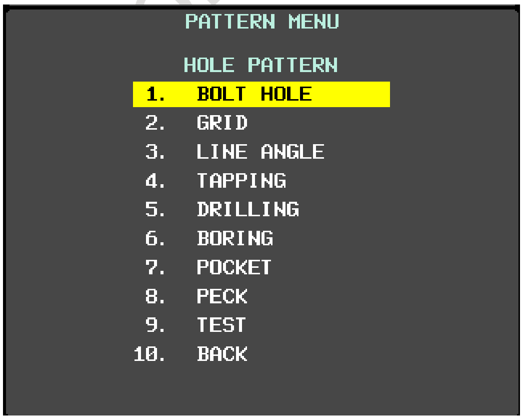


Fig. 12.21 (g) Pattern menu screen

```
O9500
N1 G65 H90 P072079 Q076069 R032080 I065084 J084069 K082078;... "HOLE PATTERN"
N2 G65 H91 P1 Q066079 R076084 I032072 J079076 K069032; ............ "BOLT HOLE"
N3 G65 H91 P2 Q071082 R073068; ....................................................... "GRID"
N4 G65 H91 P3 Q076073 R078069 I032065 J078071 K076069; ............ "LINE ANGLE"
N5 G65 H91 P4 Q084065 R080080 I073078 J071032;  ......................... "TAPPING"
N6 G65 H91 P5 Q068082 R073076 I076073 J078071;........................... "DORILLING"
N7 G65 H91 P6 Q066079 R082073 I078071; ........................................ "BORING"
N8 G65 H91 P7 Q080079 R067075 I069084; ........................................ "POCKET"
N9 G65 H91 P8 Q080069 R067075; ....................................................... "PECK"
N10 G65 H91 P9 Q084069 R083084; ..................................................... "TEST"
N11 G65 H91 P10 Q066065 R067075; ................................................... "BACK"
N12 M99;
```

Definition of the custom macro screen

The title, variable name and comment are defined as follows.

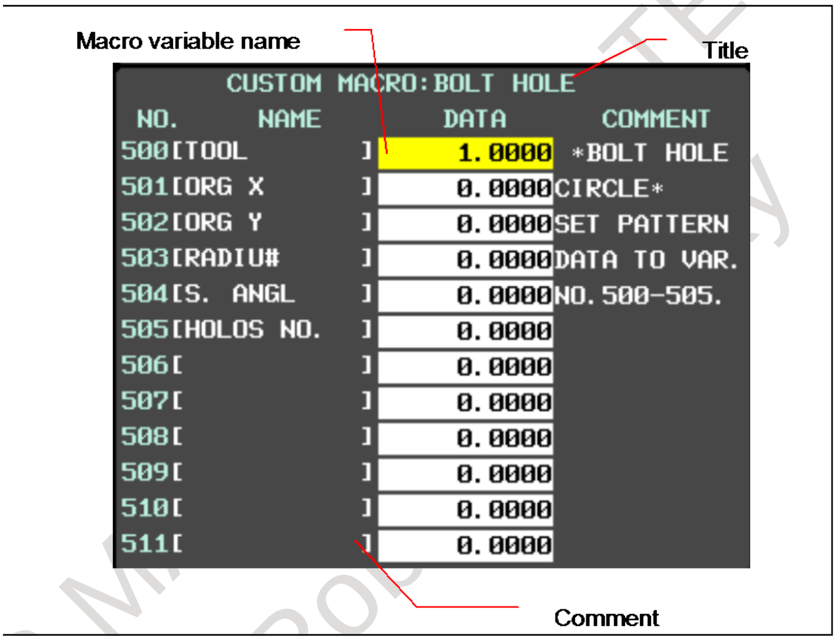


Fig. 12.21 (h) Custom macro screen

Definition of title

The character string displayed in the title of the custom macro screen is defined.
The title is specified up to 12 characters in a half size letter and up to 6 characters in a full size letter.

- Format

G65 H92 P_ Q_ R_ I_ J_ K_ ;

H92	:	Specifies the menu title
P_	:	The code of 1st and 2nd characters of the menu title
Q_	:	The code of 3rd and 4th characters of the menu title
R_	:	The code of 5th and 6th characters of the menu title
I_	:	The code of 7th and 8th characters of the menu title
J_	:	The code of 9th and 10th characters of the menu title
K_	:	The code of 11th and 12th characters of the menu title

As for the way of setting the character-code, refer to the Subsection, "Setting the Character-codes".

Definition of macro variable

The character string displayed in the macro variable name is defined.
The macro variable is specified up to 10 characters in a half size letter and up to 5 characters in a full size letter.

The following macro variable numbers can be used:
#100 to 199 (100 variables)
#500 to 999 (500 variables), 600 variables in total

- Format

G65 H93 P_ Q_ R_ I_ J_ K_ ;

H93	:	Specifies the variable name
P_	:	Specifies the variable number Specifies 100 to 199 or 500 to 999
Q_	:	The code of 1st and 2nd characters of the variable name
R_	:	The code of 3rd and 4th characters of the variable name
I_	:	The code of 5th and 6th characters of the variable name
J_	:	The code of 7th and 8th characters of the variable name
K_	:	The code of 9th and 10th characters of the variable name

As for the way of setting the character-code, refer to the Subsection, "Setting the Character-codes".

Definition of a comment

The character string of the comment displayed on the custom macro screen is defined.
The comment is specified by up to 12 characters in a half size letter and up to 6 characters in a full size letter per one block.

- When the comment is displayed at the bottom（bit 0 of parameter No. 11318=0）
  The number of blocks is defined up to maximum 9 blocks.
  One line is composition of 3 blocks and maximum 3 lines can be displayed.

  The comment is displayed in the order specified by program as the Fig. 12.21 (i).
  The block is arranged in defined order.

Block 1	Block 2	Block 3
Block 4	Block 5	Block 6
Block 7	Block 8	Block 9

Fig. 12.21 (i) Block layout of comment

- When the comment is displayed at the right (bit 0 of parameter No. 11318=1)
  1 line is composed by 1 blocks, the maximum number of lines is 8 on the 8.4-inch display unit or 12 on the 10.4-inch display unit.

- Format

G65 H94 P_ Q_ R_ I_ J_ K_ ;

H94	:	Specifies the comment
P_	:	The code of 1st and 2nd characters of comment
Q_	:	The code of 3rd and 4th characters of comment
R_	:	The code of 5th and 6th characters of comment
I_	:	The code of 7th and 8th characters of comment
J_	:	The code of 9th and 10th characters of comment
K_	:	The code of 11th and 12th characters of comment

As for the way of setting the character-code, refer to the Subsection, "Setting the Character-codes".

Example

The Fig. 12.21 (j) and Fig. 12.21 (k) are example of the custom macro screen.

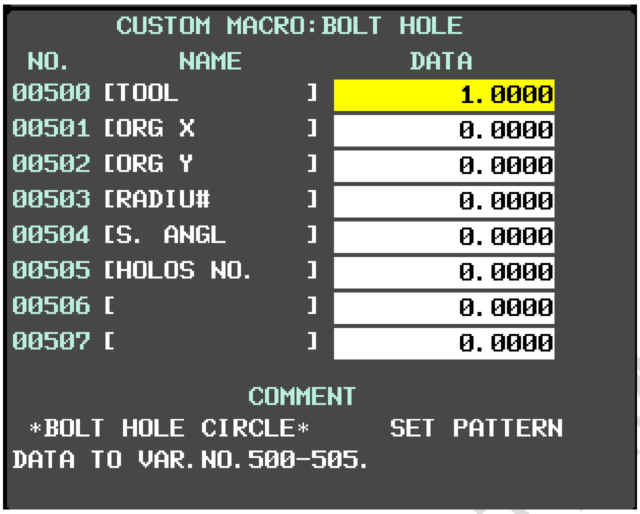


Fig. 12.21 (j) Custom macro screen（No. 11318#0=0）

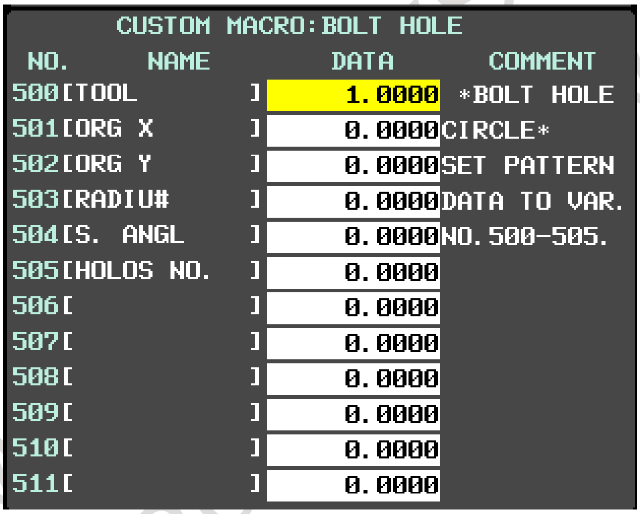


Fig. 12.21 (k) Custom macro screen（No. 11318#0=1）

```
O9501;
N1 G65 H92 P066079 Q076084 R032072 I079076 J069032;...................."BOLT HOLE"
N2 G65 H93 P500 Q084079 R079076; ........................................................"TOOL"
N3 G65 H93 P501 Q079082 R071032 I08832; ........................................."ORG X"
N4 G65 H93 P502 Q079082 R071032 I08932; ........................................."ORG Y"
N5 G65 H93 P503 Q082065 R068073 I085803; ......................................."RADIUS"
N6 G65 H93 P504 Q083046 R032065 I078071 J076032.........................."S. ANGL"
N7 G65 H93 P505 Q072079 R076079 I083032 J078079 K046032............"HOLES NO."
N8 G65 H94 P032042 Q066079 R076084 I032072 J079076 K069032;....." *BOLT HOLE"
N9 G65 H94 P067073 Q082067 R076069 I042032; ................................"CIRCLE*"
N10 G65 H94 P083069 Q084032 R080065 I084084 J069082 K078032;..."SET PATTERN"
N11 G65 H94 P068065 Q084065 R032084 I079032 J086065 K082046;..."DATA NO VAR."
N12 G65 H94 P078079 Q046053 R048048 I045053 J048053 K046032;..."NO500-505"
N13 M99;
```

Setting the character-codes

The character cannot be used to specify the NC program.
Therefore, the code corresponding to the character is specified.
One character is consist of three figures in a half size letter and six figures in a full size letter.
The character code is specified for each address of the G65 instruction by six digits.

Refer to the table for the character code for the character code.
Example)
When "ABCDEFGH" is specified, the description of the code is as follows.
Encoded character string : 065 066 067 068 069 070 071 072

<u>P065066</u> <u>Q067068</u> <u>R069070</u> <u>I071072</u> ;
AB CD EF GH

> **NOTE**
> 1 Space (032) is added ahead of the character-code, when the character-code of three digits or less is specified.
> Example)
> P065066 Q067; → " AB C "
> 032(space) is put at the end, when "ABC" is displayed.
> P065066 Q067032; → " ABC "
> 2 It is assumed in that the space of two characters was defined in the address when there is an address not defined.
> Example)
> P065066 I067068; → "AB   CD"
> 3 The character string corresponding to the block where G code other than G65 is instructed or where undefined character code is specified is not displayed.

Characters and codes to be used for the pattern data input function

Character	Code	Comment	Character	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		!	033	Exclamation mark
G	071		”	034	Quotation mark
H	072		#	035	Hash sign
I	073		$	036	Dollar sign
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		’	039	Apostrophe
M	077		*	042	Asterisk
N	078		+	043	Plus sign
O	079		,	044	Comma
P	080		-	045	Minus sign
Q	081		.	046	Period
R	082		/	047	Slash
S	083		:	058	Colon
T	084		;	059	Semicolon
U	085		<	060	Left angle bracket
V	086		=	061	Equal sign
W	087		>	062	Right angle bracket
X	088		?	063	Question mark

Character	Code	Comment	Character	Code	Comment
Y	089		@	064	At mark
Z	090		[	091	Left square bracket
0	048		¥	092	Yen sign
1	049		]	093	Right square bracket
2	050		^	094	
3	051		_	095	Underscore
4	052				
5	053				

The characters and the codes of the katakana is as follows.

Character	Code	Comment	Character	Code	Comment
ア	177		ム	209	
イ	178		メ	210	
ウ	179		モ	211	
エ	180		ヤ	212	
オ	181		ユ	213	
カ	182		ヨ	214	
キ	183		ラ	215	
ク	184		リ	216	
ケ	185		ル	217	
コ	186		レ	218	
サ	187		ロ	219	
シ	188		ワ	220	
ス	189		ヲ	166	
セ	190		ン	221	
ソ	191		ァ	167	
タ	192		ィ	168	
チ	193		ゥ	169	
ツ	194		ェ	170	
テ	195		ォ	171	
ト	196		ャ	172	
ナ	197		ュ	173	
ニ	198		ョ	174	
ヌ	199		ッ	175	
ネ	200		゛	222	voiced dot
ノ	201		゜	223	semi-voiced dot
ハ	202		。	161	Punctuation
ヒ	203		「	162	Left quotation mark
フ	204		」	163	Right quotation mark
ヘ	205		、	164	Comma
ホ	206		・	165	Point
マ	207			000	Space
ミ	208				

NOTE
Diacritical mark is one character.

The characters and the codes of the hiragana and the kanji are as follows. The following hiraganas and kanjis use two characters of the alphanumeric character.

ぁ	あ	ぃ	い	ぅ	う	ぇ	え	ぉ	お
002 000	002 002	002 004	002 006	002 008	002 010	002 012	002 014	002 016	002 018
か	が	き	ぎ	く	ぐ	け	げ	こ	ご
002 020	002 022	002 024	002 026	002 028	002 030	002 032	002 034	002 036	002 038
さ	ざ	し	じ	す	ず	せ	ぜ	そ	ぞ
002 040	002 042	002 044	002 046	002 048	002 050	002 052	002 054	002 056	002 058
た	だ	ち	ぢ	っ	つ	づ	て	で	と
002 060	002 062	002 064	002 066	002 068	002 070	002 072	002 074	002 076	002 078
ど	な	に	ぬ	ね	の	は	ば	ぱ	ひ
002 080	002 082	002 084	002 086	002 088	002 090	002 092	002 094	002 096	002 098
び	ぴ	ふ	ぶ	ぷ	へ	べ	ぺ	ほ	ぼ
002 100	002 102	002 104	002 106	002 108	002 110	002 112	002 114	002 116	002 118
ぽ	ま	み	む	め	も	ゃ	や	ゅ	ゆ
002 120	002 122	002 124	002 126	002 128	002 130	002 132	002 134	002 136	002 138
ょ	よ	ら	り	る	れ	ろ	ゎ	わ	素
002 140	002 142	002 144	002 146	002 148	002 150	002 152	002 154	002 156	002 158
材	を	ん	種	類	棒	穴	成	形	質
002 160	002 162	002 164	002 166	002 168	002 170	002 172	002 174	002 176	002 178
寸	法	外	径	長	端	面	最	小	内
002 180	002 182	002 184	002 186	002 188	002 190	002 192	002 194	002 196	002 198
大	加	工	切	削	倣	正	途	中	荒
002 200	002 202	002 204	002 206	002 208	002 210	002 212	002 214	002 216	002 218
具	番	号	仕	上	込	点	方	向	速
002 220	002 222	002 224	002 226	002 228	002 230	002 232	002 234	002 236	002 238
度	送	量	開	始	深	主	軸		
002 240	002 242	002 244	002 246	002 248	002 250	002 252	002 254		
回	転	数	位	置	決	直	線	時	円
003 000	003 002	003 004	003 006	003 008	003 010	003 012	003 014	003 016	003 018
反	現	在	指	令	値	領	域	診	断
003 020	003 022	003 024	003 026	003 028	003 030	003 032	003 034	003 036	003 038
操	作	手	引	機	械	残	移	動	次
003 040	003 042	003 044	003 046	003 048	003 050	003 052	003 054	003 056	003 058
早	電	源	投	入	間	分	秒	自	運
003 060	003 062	003 064	003 066	003 068	003 070	003 072	003 074	003 076	003 078
負	荷	実	使	用	寿	命	新	規	除
003 080	003 082	003 084	003 086	003 088	003 090	003 092	003 094	003 096	003 098
隅	取	単	補	能	独	終	了	記	角
003 100	003 102	003 104	003 106	003 108	003 110	003 112	003 114	003 116	003 118
溝	刃	幅	広	設	定	一	覧	表	部
003 120	003 122	003 124	003 126	003 128	003 130	003 132	003 134	003 136	003 138
炭	合	金	鋼	超	硬	先	付	摩	耗
003 140	003 142	003 144	003 146	003 148	003 150	003 152	003 154	003 156	003 158
仮	想	副	行	挿	消	去	山	高	準
003 160	003 162	003 164	003 166	003 168	003 170	003 172	003 174	003 176	003 178
備	完	後	弧	助	択	無	視	器	原
003 180	003 182	003 184	003 186	003 188	003 190	003 192	003 194	003 196	003 198
登	録	再	処	理	描	画	過	容	編
003 200	003 202	003 204	003 206	003 208	003 210	003 212	003 214	003 216	003 218
集	未	対	相	座	標	示	名	歯	変
003 220	003 222	003 224	003 226	003 228	003 230	003 232	003 234	003 236	003 238
呼	推	馬	力	系	選	達	閉		
003 240	003 242	003 244	003 246	003 248	003 250	003 252	003 254		

禁	復	帰	書	個	桁	稼	由	両	半
004 000	004 002	004 004	004 006	004 008	004 010	004 012	004 014	004 016	004 018
逃	底	逆	下	空	四	触	平	代	辺
004 020	004 022	004 024	004 026	004 028	004 030	004 032	004 034	004 036	004 038
格	子	周	心	本	群	停	止	巾	微
004 040	004 042	004 044	004 046	004 048	004 050	004 052	004 054	004 056	004 058
状	路	範	囲	倍	率	注	側	特	殊
004 060	004 062	004 064	004 066	004 068	004 070	004 072	004 074	004 076	004 078
距	離	連	続	増	隔	件	初	期	条
004 080	004 082	004 084	004 086	004 088	004 090	004 092	004 094	004 096	004 098
経	握	圧	扱	陰	隠	右	押	横	黄
004 100	004 102	004 104	004 106	004 108	004 110	004 112	004 114	004 116	004 118
億	屋	化	何	絵	階	概	該	巻	換
004 120	004 122	004 124	004 126	004 128	004 130	004 132	004 134	004 136	004 138
気	起	軌	技	疑	供	共	境	強	教
004 140	004 142	004 144	004 146	004 148	004 150	004 152	004 154	004 156	004 158
掘	繰	係	傾	型	検	権	研	肩	見
004 160	004 162	004 164	004 166	004 168	004 170	004 172	004 174	004 176	004 178
験	元	弦	減	孔	巧	控	更	校	構
004 180	004 182	004 184	004 186	004 188	004 190	004 192	004 194	004 196	004 198
根	左	差	雑	参	散	産	算	治	耳
004 200	004 202	004 204	004 206	004 208	004 210	004 212	004 214	004 216	004 218
式	失	修	十	従	勝	商	少	尚	昇
004 220	004 222	004 224	004 226	004 228	004 230	004 232	004 234	004 236	004 238
植	色	食	伸	信	侵	振	浸		
004 240	004 242	004 244	004 246	004 248	004 250	004 252	004 254		
真	暗	以	意	異	影	鋭	越	価	可
005 000	005 002	005 004	005 006	005 008	005 010	005 012	005 014	005 016	005 018
科	果	箇	課	各	拡	核	学	掛	漢
005 020	005 022	005 024	005 026	005 028	005 030	005 032	005 034	005 036	005 038
簡	観	関	含	却	客	休	急	業	曲
005 040	005 042	005 044	005 046	005 048	005 050	005 052	005 054	005 056	005 058
均	筋	継	計	軽	言	限	互	降	採
005 060	005 062	005 064	005 066	005 068	005 070	005 072	005 074	005 076	005 078
済	細	姿	思	写	射	斜	者	車	借
005 080	005 082	005 084	005 086	005 088	005 090	005 092	005 094	005 096	005 098
縦	重	出	述	術	渉	照	省	章	証
005 100	005 102	005 104	005 106	005 108	005 110	005 112	005 114	005 116	005 118
象	身	進	人	図	違	印	沿	遠	央
005 120	005 122	005 124	005 126	005 128	005 130	005 132	005 134	005 136	005 138
奥	往	応	会	解	改	割	活	願	基
005 140	005 142	005 144	005 146	005 148	005 150	005 152	005 154	005 156	005 158
奇	寄	岐	既	近	区	矩	駆	偶	旧
005 160	005 162	005 164	005 166	005 168	005 170	005 172	005 174	005 176	005 178
求	球	究	級	欠	結	口	語	誤	交
005 180	005 182	005 184	005 186	005 188	005 190	005 192	005 194	005 196	005 198
厚	項	刻	告	黒	財	策	糸	試	資
005 200	005 202	005 204	005 206	005 208	005 210	005 212	005 214	005 216	005 218
事	持	似	釈	弱	受	収	純	順	所
005 220	005 222	005 224	005 226	005 228	005 230	005 232	005 234	005 236	005 238
序	剰	場	常	飾	水	錐	据		
005 240	005 242	005 244	005 246	005 248	005 250	005 252	005 254		

制	整	製	前	全	然	則	属	即	他
006 000	006 002	006 004	006 006	006 008	006 010	006 012	006 014	006 016	006 018
多	存	谷	探	短	徴	鎮	調	頂	鉄
006 020	006 022	006 024	006 026	006 028	006 030	006 032	006 034	006 036	006 038
添	頭	同	導	道	熱	年	濃	箱	発
006 040	006 042	006 044	006 046	006 048	006 050	006 052	006 054	006 056	006 058
抜	伴	必	百	複	物	文	聞	併	忘
006 060	006 062	006 064	006 066	006 068	006 070	006 072	006 074	006 076	006 078
末	密	有	余	与	裏	立	略	青	席
006 080	006 082	006 084	006 086	006 088	006 090	006 092	006 094	006 096	006 098
石	積	赤	接	折	粗	創	双	捜	太
006 100	006 102	006 104	006 106	006 108	006 110	006 112	006 114	006 116	006 118
打	体	待	態	替	段	知	地	致	遅
006 120	006 122	006 124	006 126	006 128	006 130	006 132	006 134	006 136	006 138
追	通	伝	得	読	凸	凹	突	鈍	敗
006 140	006 142	006 144	006 146	006 148	006 150	006 152	006 154	006 156	006 158
杯	背	配	品	不	布	並	頁	別	片
006 160	006 162	006 164	006 166	006 168	006 170	006 172	006 174	006 176	006 178
返	勉	弁	保	明	滅	木	目	歪	揺
006 180	006 182	006 184	006 186	006 188	006 190	006 192	006 194	006 196	006 198
様	溶	要	抑	良	輪	和	話	枠	節
006 200	006 202	006 204	006 206	006 208	006 210	006 212	006 214	006 216	006 218
説	絶	千	専	浅	旋	総	走	退	台
006 220	006 222	006 224	006 226	006 228	006 230	006 232	006 234	006 236	006 238
第	題	卓	室	着	柱	鋳	丁		
006 240	006 242	006 244	006 246	006 248	006 250	006 252	006 254		
低	訂	肉	日	白	薄	比	皮	被	非
007 000	007 002	007 004	007 006	007 008	007 010	007 012	007 014	007 016	007 018
美	普	伏	歩	包	門	問	絡	列	万
007 020	007 022	007 024	007 026	007 028	007 030	007 032	007 034	007 036	007 038
利	訳	礼	乱	放	枚	約	練	油	劣
007 040	007 042	007 044	007 046	007 048	007 050	007 052	007 054	007 056	007 058
例	郭	戻	冷	垂	緑	紫	許	測	精
007 060	007 062	007 064	007 066	007 068	007 070	007 072	007 074	007 076	007 078
効	→	↗	↑	↖	←	↙	↓	↘	
007 080	007 082	007 084	007 086	007 088	007 090	007 092	007 094	007 096	007 098
				板	予	〃	家	装	管
007 100	007 102	007 104	007 106	007 108	007 110	007 112	007 114	007 116	007 118
粉	等					貫	安	α	β
007 120	007 122	007 124	007 126	007 128	007 130	007 132	007 134	007 136	007 138
程	抗	張	任	破	損	御	足	守	般
007 140	007 142	007 144	007 146	007 148	007 150	007 152	007 154	007 156	007 158
納	義	丸	汎	固	毎	当	的	詳	鳥
007 160	007 162	007 164	007 166	007 168	007 170	007 172	007 174	007 176	007 178
適	論	額	縁	温	給	界	混	監	締
007 180	007 182	007 184	007 186	007 188	007 190	007 192	007 194	007 196	007 198
護	己	称	樹	脂	料	落	確	認	報
007 200	007 202	007 204	007 206	007 208	007 210	007 212	007 214	007 216	007 218
排	性	生	績	判	搬	砥	θ	島	壁
007 220	007 222	007 224	007 226	007 228	007 230	007 232	007 234	007 236	007 238
				]	[	╻	■		
007 240	007 242	007 244	007 246	007 248	007 250	007 252	007 254		

12.22 G CODE PREVENTING BUFFERING

Overview

By specifying G04.1, buffering of the following block from the block of G04.1 is prevented until finishing the block of G04.1. (At the following, preventing buffering by G04.1 is called non-buffering command by G code. On the other hand, preventing buffering by M codes, command of only G31, command of only G53 and etc. are called non-buffering commands of normal.)

The processing time of non-buffering command by G code is reduced compared with non-buffering commands of normal. Also, the position of releasing preventing buffering can be specified by the distance to go in the previous movement block. Therefore, the cycle time of automatic operation can be reduced by using non-buffering command by G code instead of non-buffering commands of normal.

Specification

By specifying G04.1, the block which includes G04.1 becomes the prevented block of buffering.

Format

G04.1 (P_) ;

P_ : The operation mode of G04.1.
Compatible operation to preventing buffering by command of only G31 or G53 is possible when P1 or P2 is specified with G04.1 in the same block.
P1: Compatible operation to preventing buffering by command of only G31.
P2: Compatible operation to preventing buffering by command of only G53.

However, alarm PS2085, "G04.1 FORMAT ERROR" occurs when P code except P1 and P2 is specified.

I_ : Distance to go for releasing preventing buffering (Synthetic distance to go in all axis)

However, alarm (PS2085) occurs when the address I is specified in the state of setting the bit 5 (MRR) of parameter No.11279 to 0.

NOTE

1. When P1 is specified, the system variables (#100151 to #100182) of skip position are updated by executed position of the block that includes G04.1. (When the system up to 20 axes, the system variables (#5061 to #5080) compatible with FS16 is uploaded too.)
2. About preventing buffering by M codes or command of only G31 or command of only G53, those behaviors are different respectively. When each preventing buffering is exchanged for G04.1, it is necessary to use the compatible operation mode.
3. Unit system as for the address I depends on the reference axis.
4. The address I is commanded by radius value regardless of diameter/radius specification of the reference axis.
5. Positive value should be specified as for the address I. If negative value is specified, the command of the address I is invalid.
6. The command of address I is invalid when the previous block of G04.1 is not the movement command.

Explanation

About G04.1

G04.1 is a one-shot G code. When the commands except sequence number, P codes and comment are specified with G04.1 in the same block, alarm PS2085, “G04.1 FORMAT ERROR” occurs.

About processing time

Because the processing time of non-buffering command by G code is reduced compared with non-buffering commands of normal, the execution time of the block which includes non-buffering command is reduced.

Speed-up of non-buffering command by G code invalid signal NHSW<G0579.6>

Speed-up of non-buffering command by G code is invalid when Speed-up of non-buffering command by G code invalid signal NHSW<G0579.6> is set to “1”. And the processing time of non-buffering command by G code is equal with non-buffering commands of normal.

Suppressing single block stop

Single block stop is not performed in G04.1 block when bit 6 (MSB) of parameter No.11279 is set to 1.
In this case, the function of preventing buffering in G04.1 block is not invalid.

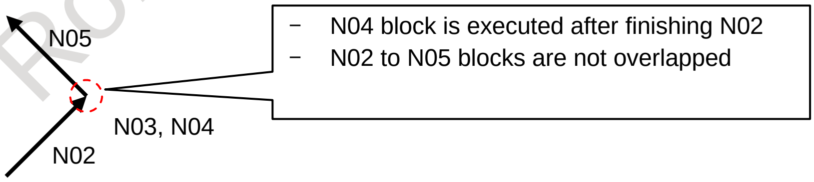
> **NOTE**
> Single block stop is not performed in G04.1 block regardless of the state of #3003 when bit 6 (MSB) of parameter No.11279 is set to 1.

When parameter MMR is set to 0

If G04.1 is specified just behind the movement block, the buffering of the block behind G04.1 is started when the movement block finishes and each axis reaches to in-position.

Example)
In case of the following program, the buffering of the block behind N03 is started when the movement command of N02 finishes and each axis reaches to in-position (N02 block finishes).
Therefore, the macro statement of N04 is executed after finishing N02.

```
O0001
N01 G90 G00 X0 Y0
N02 G01 X100.0 Y100.0 F1000.0
N03 G04.1
N04 #100=#101+#102
N05 G01 X0.0 Y200.0 F1000.0
…
```



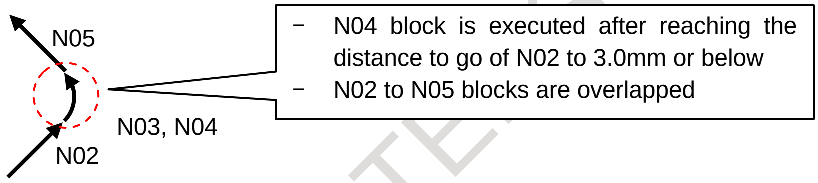
And, even if the following command is specified in the blocks which are ahead or behind G04.1 block, the movement commands are not overlapped.
- Cutting feed – Cutting feed block
- Rapid traverse – Rapid traverse block with rapid traverse block overlap
- Movement block – Movement block with smart overlap

When parameter MMR is set to 1

If G04.1 is specified just behind the movement block, the buffering of the block behind G04.1 is started when the distance to go of the previous movement block reaches to the specified distance to go (distance to go is specified by address I).

Example)
In case of the following program is executed, the buffering of the block behind N03 is started when the distance to go of the movement command of N02 reaches to the distance to go (distance to go is specified by address I).
Therefore, the macro statement of N04 is executed after reaching the distance to go of N02 to 3.0mm or below.

```
O0002
N01 G90 G00 X0 Y0
N02 G01 X100.0 Y100.0 F1000.0
N03 G04.1 I3.0
N04 #100=#101+#102
N05 G01 X0.0 Y200.0 F1000.0
…
```



And, if the following command is specified in the blocks which are ahead or behind G04.1 block, the movement commands are overlapped.

- Cutting feed – Cutting feed block
- Rapid traverse – Rapid traverse block with rapid traverse block overlap
- Movement block – Movement block with smart overlap

NOTE
1. The delay occurs from starting the buffering of the block behind G04.1 to executing it. And, the delay time is uneven.
2. When the distance to go by the address I is short, the movement command might be not overlapped.

Limitation

Display of the currently executing program

The block of non-buffering command by G code is not displayed in the currently executing program.
However, the block of non-buffering command by G code is displayed in the currently executing program when speed-up is invalid by the condition which is described in “Condition to treat non-buffering command by G code as non-buffering commands of normal”.

Condition to treat non-buffering command by G code as non-buffering commands of normal

When one of the following conditions is satisfied, speed-up of non-buffering command by G code is invalid, and the processing time of non-buffering command by G code is equal with non-buffering commands of normal. Moreover, non-buffering command by G code is treated as non-buffering commands of normal.

- Speed-up of non-buffering command by G code invalid signal is set to “1”.
- The single-block operation is enabled.
- Non-buffering command by G code is specified in the same block as the program number.
- In manual handle retrace function, the checking mode synchronizing an operation with a manual handle pulse is selected.
- Animation or path drawing of dynamic graphic display is operated.
- Bit 7(HMA) of parameter No.11279 is set to 1, and non-buffering command by G code is specified during Relation of cutter compensation, Multiple repetitive canned cycle G70 to G73.

Manual handle retrace function

In Manual handle retrace function, a backward movement cannot be executed while the block of non-buffering command by G code is executing except the checking mode synchronizing with a manual handle pulse. The backward movement ends when an operation enters the next block of non-buffering command by G code during the backward movement. And ‘RVED’ is displayed.
However, in the checking mode synchronizing with a manual handle pulse, the backward movement can be executed when non-buffering command by G code was executed once during forward movement, because the non-buffering command by G code is treated as non-buffering commands of normal in this case.

Retrace

The backward movement cannot be executed by Retrace function in the block of non-buffering command by G code. The backward movement ends when the next block of non-buffering command by G code just starts executing during the backward movement. And ‘RVED’ is displayed.

Relation of cutter compensation

Non-buffering command by G code cannot be specified during the following cutter compensation functions. Alarm PS0010, “IMPROPER G-CODE” occurs when such a command is specified. Use non-buffering commands of normal during the following cutter compensation functions, or set bit 7 (HMA) of parameter No.11279 to 1.

- Cutter compensation and tool nose radius compensation

> **NOTE**
> Non-buffering command by G code cannot be specified just after the cancel command of cutter compensation, tool nose radius compensation. Moreover, non-buffering command by G code cannot be specified while the vector of cutter compensation is being held.

T

Multiple repetitive canned cycle

Non-buffering command by G code cannot be specified during Multiple repetitive canned cycle G70 to G73. Alarm PS0010 occurs when such a command is specified. Use non-buffering commands of normal during Multiple repetitive canned cycle G70 to G73, or set bit 7 (HMA) of parameter No.11279 to 1.

Chamfering and corner R, Optional chamfering and corner R and Direct drawing dimension programming

In the condition to treat non-buffering command by G code as non-buffering commands of normal, alarm PS0051, “MISSING MOVE AFTER CNR/CHF” occurs when non-buffering command by G code is specified during Chamfering and corner R or Optional chamfering and corner R. And alarm PS0312, “ILLEGAL COMMAND IN DIRECT DRAWING DIMENSIONS PROGRAMMING”occurs when non-buffering command by G code is specified during Direct drawing dimension programming.

Block start interlock signal

Block start interlock signal *BSL<Gn008.3> is invalid for non-buffering command by G code. However, block start interlock signal *BSL<Gn008.3> is valid in the condition to treat non-buffering command by G code as non-buffering commands of normal.

When the speed-up is invalid

When the speed-up for G04.1 is invalid, the address I is ignored even if the address I is specified.
Example for the ignoring)

- Speed-up of non-buffering command by G code invalid signal NHSW<G0579.6> is set to “1”
- Single block operation

Feed hold stop

When a feed hold stop is operated in the previous block of G04.1, the operation of G04.1 after restart is changed according to the number of blocks read ahead.

<When the number of blocks read ahead is 1>

The address I is ignored.

<When the number of blocks read ahead is more than 2>

The address I is not ignored.

When a feed hold stop is operated in the previous block of G04.1, and the number of blocks read ahead is 1, the address I in the block of G04.1 is ignored.

Synchronous, composite, and superimposed control, Axis synchronous control, Flexible synchronization control

When the target axis of Synchronous control, Superimposed control, Axis synchronous control or Flexible synchronization control exists in the path, release of preventing buffering by distance to go is operated as follows.

Table 12.22 (a) Movement of master and slave axis

Command	Movement of master axis	Movement of slave axis
Synchronous control	Added for distance to go	Excluded from distance to go
Superimposed control		
Axis synchronous control		
Flexible synchronization control		

And, the movement of the target axis of Composite control is added for distance to go.

The functions that cannot be specified

Release of preventing buffering by distance to go cannot be used when the movement commands (rapid traverse (G00), linear interpolation (G01), circular interpolation (G02, G03), helical interpolation (G02, G03)) are not specified behind G04.1 block.

However, even if the above movement command is specified, release of preventing buffering by distance to go cannot be used in the state as follows.

- G04.1 block is specified on just behind the commands of following table.
- G04.1 block is specified on the mode of the commands of following table.

When release of preventing buffering by distance to go cannot use, the address I is ignored even if the address I is specified.

Table 12.22 (b) The commands that release of preventing buffering by distance to go cannot be used

- : Nothing

Command	Just behind	mode
Dwell (G04)	Cannot use	-
Exact stop (G09)	Cannot use	-
Exact stop mode (G61)	Cannot use	Cannot use
Programmable data input (G10)	Cannot use	Cannot use
Programmable parameter input (G10)	Cannot use	Cannot use
Polar coordinates command (G15, G16)	Cannot use	Cannot use
Inch/metric conversion (G20, G21)	Cannot use	-
Reference position return check (G27)	Cannot use	-
Reference position return (G28)	Cannot use	-
2nd Reference position return (G30P2)	Cannot use	-
3rd and 4th Reference position return (G30P3, G30P4)	Cannot use	-
In-position check disable reference position return (G28.2, G30.2P2, G30.2P3, G30.2P4)	Cannot use	-

Command	Just behind	mode
Skip function (G31)	Cannot use	-
Multi-step skip (G31P1 to P4)	Cannot use	-
High-speed skip (G31)	Cannot use	-
Continuous high-speed skip function (G31P90)	Cannot use	-
Torque limit skip (G31P99)	Cannot use	-
Threading (G32)	Cannot use	Cannot use
Continuous threading (G32)	Cannot use	Cannot use
Multi threading (G32)	Cannot use	Cannot use
Variable lead thread cutting (G34)	Cannot use	Cannot use
Circular thread cutting (G35, G36)	Cannot use	Cannot use
Circular thread cutting B (G02.1, G03.1)	Cannot use	Cannot use
Arbitrary speed threading (G31)	Cannot use	Cannot use
Automatic tool length measurement, Automatic tool offset (G36, G37)	Cannot use	-
Normal direction control (G41.1, G42.1, G40.1)	Cannot use	Cannot use
Synchronous, composite, and superimposed control by program command (G50.4, G51.4, G50.5, G51.5, G50.6, G51.6)	Cannot use	Cannot use
Polygon turning (G51.2, G50.2)	Cannot use	Cannot use
Polygon machining with two spindles (G51.2, G50.2)	Cannot use	Cannot use
Setting a workpiece coordinate system (G92(M) / G50(T))	Cannot use	-
Workpiece coordinate system preset (G92.1(M) / G50.3(T))	Cannot use	-
Local coordinate system (G52)	Cannot use	-
Flexible path axis assignment (G52.1, G52.2, G52.3)	Cannot use	-
Workpiece coordinate system (G54～G59)	Cannot use	-
3-dimensional coordinate system conversion (G68, G69)	Cannot use	Cannot use
Tilted working plane indexing (G68.2, G69)	Cannot use	Cannot use
Electronic gear box (G80, G81)	Cannot use	Cannot use
Electronic gear box 2 pair (G80.5, G81.5)	Cannot use	Cannot use
Rigid tapping (G84)	Cannot use	Cannot use
Torch swing	-	Cannot use

Signal

Speed-up of non-buffering command by G code invalid signal NHSW<G0579.6>

[Classification] Input signal

[Function] Speed-up of non-buffering command by G code is invalidated.

[Operation] When this signal becomes “1”:

Speed-up of non-buffering command by G code is invalidated, and non-buffering command by G code is treated as non-buffering commands of normal.

When this signal becomes “0”:

Speed-up of non-buffering command by G code is validated.

> **NOTE**
> This signal cannot be changed during operating a program. The state of signal at cycle start is kept.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0579		NHSW						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11279	HMA	MSB	MRR					

[Input type] Parameter input
[Data type] Bit

#5 MRR In G code preventing buffering (G04.1), the position of releasing preventing buffering:
0: Cannot be specified by the distance to go in the previous movement block.
1: Can be specified by the distance to go in the previous movement block.

#6 MSB When single block signal SBK <Gn046.1> is set to "1", single block stop in G code preventing buffering (G04.1) is:
0: performed.
1: not performed.

NOTE
Single block stop is not performed in G04.1 block regardless of the state of #3003 when bit 6 (MSB) of parameter No.11279 is set to 1.

#7 HMA When non-buffering command by G code is specified during Relation of cutter compensation, Multiple repetitive canned cycle G70 to G73, :
0: alarm occurs.
1: alarm not occurs.

NOTE
1 Processing time of non-buffering command by G code during Relation of cutter compensation, Multiple repetitive canned cycle G70 to G73 is equal with non-buffering commands of normal when setting this parameter to 1.
2 Non-buffering command by G code during " Relation of cutter compensation, Multiple repetitive canned cycle G70 to G73" is treated with non-buffering commands of normal when setting this parameter to 1. Please note enough about the function which changes operation by the non-movement block (For example, Cutter compensation and Tool nose radius compensation or etc.)

Alarm and message

Number	Message	Description
PS0010	IMPROPER G-CODE	Non-buffering command by G code was specified during/just behind the command of relation of cutter compensation.
PS0051	MISSING MOVE AFTER CNR/CHF	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.
PS0312	ILLEGAL COMMAND IN DIRECT DRAWING DIMENSIONS PROGRAMMING	Direct input of drawing dimensions was commanded in an invalid format. An attempt was made to specify an invalid G code during direct input of drawing dimensions. Two or more blocks not to be moved exist in consecutive commands that specify direct input of drawing dimensions. Although non-use of commas (,) (bit 4 of parameter No. 3405 = 1) was specified for direct input of drawing dimensions, a comma was specified.
PS2085	G04.1 FORMAT ERROR	G04.1 block is not specified singly.

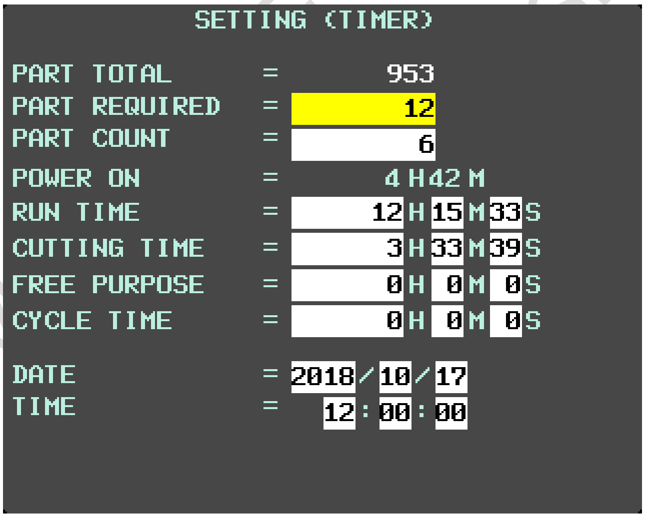
13 DISPLAY/SET/EDIT

13.1 DISPLAY/SET

13.1.1 Run Hour and Parts Count Display

Overview

This function displays the integrated power-on time, the integrated cycle operation time, the integrated cutting time and timer (started by an input signal from PMC) on the screen. The integrated cycle operation time, the integrated cutting time and timer can be altered and preset, by the operator.
In addition to the above, this function displays the count of the total number of parts machined, the number of parts required and the number of completed parts on the screen. Each time M02, M30 or a parameter set M code is executed, the count of the total number of parts machined and the number of parts completed is incremented by 1.
If a program is prepared so as to execute M02, M30 or a parameter set M code each time one part machining is completed, the number of parts machined can be counted automatically.
If the count of the number of parts machined reaches the number of parts required, a signal is output to the PMC side.
It is possible for the operator to change and preset the number of parts required and the number of parts completed.



Signal

General-purpose integrating meter start signal TMRON<Gn053.0>

[Classification] Input signal

[Function] The each path of CNC has an meter which is started by an input signal from the PMC. Additionally, there are meters for counting the automatic operation time and counting cutting time. The count for these meter can be displayed on the screen. The count can be preset by the operator.

[Operation] When the signal is set to 1, the meter starts counting.

Target part count reached signal PRTSF<Fn062.7>

[Classification] Output signal

[Function] Reports to the PMC that the specified number of parts have been machined.

[Output cond.] The PRTSF signal is set to 1 when:

- The number of parts machined counts up and reaches the required number of parts when M02, M30, or the M code set in parameter No. 6710 is executed.
  When the required number of parts is zero, this signal is not set.

The PRTSF signal is set to 0 when:

- Machining of the specified number of parts has not yet been completed.
- The system is reset.
  When PRT (bit 1 (PRT) of parameter No. 6700) is set to 1, however, the PRTSF signal is not set to 0 even if the system is reset.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn053								TMRON
Fn062	PRTSF							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6700							PRT	PCM

[Input type] Parameter input
[Data type] Bit path

#0 PCM M code that counts the total number of machined parts and the number of machined parts

0: M02, or M30, or an M code specified by parameter No. 6710
1: Only M code specified by parameter No. 6710

#1 PRT Upon reset, the required parts count arrival signal PRTSF <Fn062.7> is:

0: Set to 0.
1: Not set to 0.

6710	M code that counts the number of machined parts

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 999999999

The total number of machined parts and the number of machined parts are counted (+1) when the M code set is executed.

> **NOTE**
> The setting of 0 is invalid (no count operation is performed with M00.) Moreover, M98, M99, M198 (external device subprogram calling), and M codes used for subprogram calling and macro calling cannot be set as M codes for count-up operation. (Even when such an M code is set, count-up operation is not performed, ignoring the M code.)

6711	Number of machined parts

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 999999999

The number of machined parts is counted (+1) together with the total number of machined parts when the M02, M30, or a M code specified by parameter No. 6710 is executed.

> **NOTE**
> The number of parts is not counted for M02, M30 when bit 0 (PCM) of parameter No. 6700 is set to 1.

6712	Total number of machined parts

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 999999999

This parameter sets the total number of machined parts.
The total number of machined parts is counted (+1) when M02, M30, or an M code specified by parameter No. 6710 is executed.

> **NOTE**
> The number of parts is not counted for M02, M30, when bit 0 (PCM) of parameter No. 6700 is set to 1.

6713	Number of required parts

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 999999999

This parameter sets the number of required machined parts.
Required parts finish signal PRTSF <F0062.7> is output to PMC when the number of machined parts reaches the number of required parts. The number of parts is regarded as infinity when the number of required parts is zero. The PRTSF signal is then not output.

6750	Integrated value of power-on period

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min
[Valid data range] 0 to 999999999

This parameter displays the integrated value of power-on period.

6751	Operation time (integrated value of time during automatic operation) 1

[Input type] Setting input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 59999

6752	Operation time (integrated value of time during automatic operation) 2

[Input type] Setting input
[Data type] 2-word path
[Unit of data] min
[Valid data range] 0 to 999999999

This parameter displays the integrated value of time during automatic operation (neither stops nor hold time included).
The actual time accumulated during operation is the sum of this parameters Nos. 6751 and 6752.

6753	Integrated value of cutting time 1

[Input type] Setting input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 59999

6754	Integrated value of cutting time 2

[Input type] Setting input
[Data type] 2-word path
[Unit of data] min
[Valid data range] 0 to 999999999

This parameter displays the integrated value of a cutting time that is performed in cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03).
The actual time accumulated during cutting is the sum of this parameters Nos. 6753 and 6754.

6755	Integrated value of general-purpose integrating meter drive signal (TMRON) ON time 1

[Input type] Setting input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 59999

For details, see the description of parameter No. 6756.

6756	Integrated value of general-purpose integrating meter drive signal (TMRON) ON time 2

[Input type] Setting input
[Data type] 2-word path
[Unit of data] min
[Valid data range] 0 to 999999999

This parameter displays the integrated value of time during which General-purpose integrating meter start signal TMRON <Gn053.0> from PMC is “1”.
The actual integrated time is the sum of this parameters Nos. 6755 and 6756.

6757	Operation time (integrated value of one automatic operation time) 1

[Input type] Setting input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 59999

For details, see the description of parameter No. 6758.

6758	Operation time (integrated value of one automatic operation time) 2

[Input type] Setting input
[Data type] 2-word path
[Unit of data] min
[Valid data range] 0 to 999999999

This parameter displays the one automatic operation drive time (neither stop nor hold state included). The actual time accumulated during operating is the sum of this parameters Nos. 6757 and 6758. The operation time is automatically preset to 0 during the power-on sequence and the cycle start from the reset state.

	#7	#6	#5	#4	#3	#2	#1	#0
11651	DCO							

[Input type] Parameter input
[Data type] Bit path

#7 DCO During dry run, the cutting time is:
0: Not counted.
1: Counted.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Displaying and setting run time, parts count, and time

13.1.2 Software Operator's Panel

Overview

The MDI unit can replace the switches on the machine operator's panel. That is, the MDI unit can select a mode or jog feed override, omitting the corresponding switches on the machine operator's panel.

The control switches for the functions listed in the following table can be replaced with soft switches. Also available are 16 general-purpose soft switches which can be assigned by the machine tool builder. These 16 general-purpose soft switches can be optionally named by the machine tool builder. For control switches in groups 1 to 7, parameter can be used to select whether the control switches on the machine operator's panel or soft switches on the MDI of the control unit are used.

Group1 : Mode selection
Group2 : Selection of jog feed axis, manual rapid traverse
Group3 : Selection of manual pulse generator feed axis, selection of manual pulse magnification
Group4 : Jog feedrate override, feedrate override, rapid traverse override
Group5 : Optional block skip, single block, machine lock, dry run
Group6 : Protect key
Group7 : Feed hold
Group8 : General purpose

The states of all soft switches are input to the PMC by output signals. Based on these output signals, the PMC should turn 1 or 0 input signals related to the soft switch functions.
When the soft switch provided for single block operation is turned on, for instance, the control unit does not select the single block operation internally. The single block operation is selected just when the PMC sets the input signal for single block operation to 1.

Signal

Group	Function	Output signal		Related input signal
1	Mode selection	MD1O MD2O MD4O ZRNO	<Fn073.0> <Fn073.1> <Fn073.2> <Fn073.4>	MD1 MD2 MD4 ZRN
2	Jog feed axis select	+J1O to +J4O -J1O to -J4O	<Fn081>	+J1 to +J4 -J1 to -J4
	Manual rapid traverse	RTO	<Fn077.6>	RT
3	Manual pulse generator feed axis select	HS1AO HS1BO HS1CO HS1DO	<Fn077.0> <Fn077.1> <Fn077.2> <Fn077.3>	HS1A HS1B HS1C HS1D
	Manual pulse generator magnification rate select	MP1O MP2O	<Fn076.0> <Fn076.1>	MP1 MP2
4	Jog feed rate override	*JV0O to *JV15O	<Fn079, Fn080>	*JV0 to *JV15
	Feedrate override	*FV0O to *FV7O	<Fn078>	*FV0 to *FV7
	Rapid traverse override	ROV1O ROV2O	<Fn076.4> <Fn076.5>	ROV1 ROV2
5	Optional block skip	BDTO	<Fn075.2>	BDT
	Single block	SBKO	<Fn075.3>	SBK
	Machine lock	MLKO	<Fn075.4>	MLK
	Dry run	DRNO	<Fn075.5>	DRN
6	Protect key	KEYO*1	<F0075.6>	KEY1 to KEY4
7	Feed hold	SPO	<Fn075.7>	*SP
8	General purpose switch 1 to 8	OUT0 to OUT7	<Fn072>	
	General purpose switch 9 to 16	OUT8 to OUT15	<Fn074>	

*1 : For a multi-path system, the memory protect signal KEYO is KEYO<F0075.6>. This signal is common to all paths.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn072	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
Fn073				ZRNO		MD4O	MD2O	MD1O
Fn074	OUT15	OUT14	OUT13	OUT12	OUT11	OUT10	OUT9	OUT8
Fn075	SPO	KEYO*1	DRNO	MLKO	SBKO	BDTO		
Fn076			ROV2O	ROV1O			MP2O	MP1O
Fn077		RTO			HS1DO	HS1CO	HS1BO	HS1AO
Fn078	*FV7O	*FV6O	*FV5O	*FV4O	*FV3O	*FV2O	*FV1O	*FV0O
Fn079	*JV7O	*JV6O	*JV5O	*JV4O	*JV3O	*JV2O	*JV1O	*JV0O
Fn080	*JV15O	*JV14O	*JV13O	*JV12O	*JV11O	*JV10O	*JV9O	*JV8O
Fn081	-J4O	+J4O	-J3O	+J3O	-J2O	+J2O	-J1O	+J1O

*1 : For a multi-path system, the memory protect signal KEYO is KEYO<F0075.6>. This signal is common to all paths.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7200		OP7	OP6	OP5	OP4	OP3	OP2	OP1

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 OP1 Mode selection on software operator's panel
0: Not performed
1: Performed

#1 OP2 JOG feed axis select and manual rapid traverse select on software operator's panel
0: Not performed
1: Performed

#2 OP3 Manual pulse generator's axis select and manual pulse generator's magnification select on software operator's panel
0: Not performed
1: Performed

#3 OP4 JOG feedrate override select, feedrate override select, and rapid traverse override select on software operator's panel
0: Not performed
1: Performed

#4 OP5 Optional block skip select, single block select, machine lock select, and dry run select on software operator's panel
0: Not performed
1: Performed

#5 OP6 Protect key on software operator's panel
0: Not performed
1: Performed

#6 OP7 Feed hold on software operator's panel
0: Not performed
1: Performed

	#7	#6	#5	#4	#3	#2	#1	#0
7201								JPC

[Input type] Parameter input
[Data type] Bit path

#0 JPC To the name of a general-purpose switch on the software operator's panel , the full-size character such as Chinese character
0: can not be set.
1: can be set.

	#7	#6	#5	#4	#3	#2	#1	#0
3002				**IOV**				

[Input type] Parameter input
[Data type] Bit path

#4 IOV Override-related signal logic is:

0: Used without modification
(A signal of negative logic is used as a negative logic signal, and a signal of positive logic is used as a positive logic signal.)

1: Inverted
(A signal of negative logic is used as a positive logic signal, and a signal of positive logic is used as a negative logic signal.)

The signals indicated below are affected.

Signal of negative logic:
- Feedrate override signals *FV0 to *FV7<Gn012>
- Second feedrate override signals*AFV0 to *AFV7<Gn013>
- Feedrate override signals (for PMC axis control)
  *EFOV0g to *EFOV7g<G0151/G0163/G0175/G0187>
- Software operator’s panel signals *FV0O to *FV7O<Fn078>

Signals of positive logic:
- Rapid traverse override signals ROV1,ROV2<Gn014.0, Gn014.1>
- Software operator’s panel signals ROV1O,ROV2O<Fn076.4, Fn076.5>
- Rapid traverse override signals (for PMC axis control)
  EROV1g,EROV2g<G0150.0, G0150.1/G0162.0, G0162.1/G0174.0, G0174.1/G0186.0, G0186.1>

The signals indicated below are not affected.
- 1% step rapid traverse override selection signal HROV <Gn096.7>
- 1% step rapid traverse override signals *HROV0 to *HROV6 <Gn096.0 to Gn096.6>
- 0.1% step rapid traverse override selection signal FHROV <Gn353.7>
- 0.1% step rapid traverse override signals *FHRO0 to *FHRO9 <Gn352.0 to Gn352.7, Gn353.0 to Gn353.1>

7210	**Jog-movement axis and its direction on software operator's panel “↑”**
7211	**Jog-movement axis and its direction on software operator's panel “↓”**
7212	**Jog-movement axis and its direction on software operator's panel “→”**
7213	**Jog-movement axis and its direction on software operator's panel “←”**
7214	**Jog-movement axis and its direction on software operator's panel “↙”**
7215	**Jog-movement axis and its direction on software operator's panel “↗”**
7216	**Jog-movement axis and its direction on software operator's panel “⤴”**
7217	**Jog-movement axis and its direction on software operator's panel “↷”**

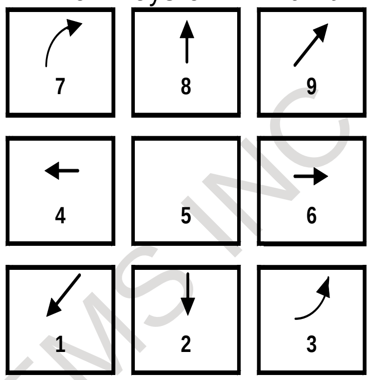
[Input type] Parameter input
[Data type] Byte path

[Valid data range] 0 to 8

On software operator's panel, set a feed axis corresponding to an arrow key on the MDI unit when jog feed is performed.

Setting value	Feed axis and direction
0	Not moved
1	First axis, positive direction
2	First axis, negative direction
3	Second axis, positive direction
4	Second axis, negative direction
5	Third axis, positive direction
6	Third axis, negative direction
7	Fourth axis, positive direction
8	Fourth axis, negative direction

Arrow keys on MDI unit



[Example] Under X, Y, and Z axis configuration, to set arrow keys to feed the axes in the direction specified as follows, set the parameters to the values given below. <8↑> to the positive direction of the Z axis, <2↓> to the negative direction of the Z axis, <6→> to the positive direction of the X axis <4←> to the negative direction of the X axis, <1↙> to the positive direction of the Y axis, <9↗> to the negative direction of the Y axis

Parameter No. 7210 = 5 (Z axis, positive direction)
Parameter No. 7211 = 6 (Z axis, negative direction)
Parameter No. 7212 = 1 (X axis, positive direction)
Parameter No. 7213 = 2 (X axis, negative direction)
Parameter No. 7214 = 3 (Y axis, positive direction)
Parameter No. 7215 = 4 (Y axis, negative direction)
Parameter No. 7216 = 0 (Not used)
Parameter No. 7217 = 0 (Not used)

7220 **Name of general-purpose switch 1 on software operator's panel (first half-size character)**
to **to**
7283 **Name of general-purpose switch 8 on software operator's panel (eighth half-size character)**

7284 **Name of general-purpose switch 9 on software operator's panel (first half-size character)**
to **to**
7299 **Name of general-purpose switch 10 on software operator's panel (eighth half-size character)**

7352 **Name of general-purpose switch 11 on software operator's panel (first half-size character)**
to **to**
7399 **Name of general-purpose switch 16 on software operator's panel (eighth half-size character)**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] -128 to 127

Each of these parameters sets the name of a general-purpose switch on the software operator's panel with the following character codes.

When bit 0 (JPC) of parameter No. 7201 is set to 0, the character codes in the “Half-size character code list” can be specified.
When bit 0 (JPC) of parameter No. 7201 is set to 1, the character codes in the “Half-size character code list” and the “Full-size character code list (a) to (f)” can be specified. The full-size character which uses two characters of the half-size can be specified by the high-code and the low-code.

A switch name consists of up to eight half-size characters (or four full-size characters).

Parameters Nos. 7220 to 7227 : Name of general-purpose switch 1
Parameters Nos. 7228 to 7235 : Name of general-purpose switch 2
Parameters Nos. 7236 to 7243 : Name of general-purpose switch 3
Parameters Nos. 7244 to 7251 : Name of general-purpose switch 4
Parameters Nos. 7252 to 7259 : Name of general-purpose switch 5
Parameters Nos. 7260 to 7267 : Name of general-purpose switch 6
Parameters Nos. 7268 to 7275 : Name of general-purpose switch 7
Parameters Nos. 7276 to 7283 : Name of general-purpose switch 8
Parameters Nos. 7284 to 7291 : Name of general-purpose switch 9
Parameters Nos. 7292 to 7299 : Name of general-purpose switch 10
Parameters Nos. 7352 to 7359 : Name of general-purpose switch 11
Parameters Nos. 7360 to 7367 : Name of general-purpose switch 12
Parameters Nos. 7368 to 7375 : Name of general-purpose switch 13
Parameters Nos. 7376 to 7383 : Name of general-purpose switch 14
Parameters Nos. 7384 to 7391 : Name of general-purpose switch 15
Parameters Nos. 7392 to 7399 : Name of general-purpose switch 16

Ex) Setting of the name of general-purpose switch 1 on the software operator's panel
The name of general-purpose switch 1 on the software operator's panel can be set to "OVR信号1", in the following setting of the parameters Nos. 7220 to 7227.
(When bit 0 (JPC) of parameter No. 7201 is set to 1)

<table>
<tr><th>Parameter</th><th>Setting value</th><th>Character</th></tr>
<tr><td>7220</td><td>79</td><td>O</td></tr>
<tr><td>7221</td><td>86</td><td>V</td></tr>
<tr><td>7222</td><td>82</td><td>R</td></tr>
<tr><td>7223</td><td>4</td><td rowspan="2">信</td></tr>
<tr><td>7224</td><td>-8</td></tr>
<tr><td>7225</td><td>2</td><td rowspan="2">号</td></tr>
<tr><td>7226</td><td>-32</td></tr>
<tr><td>7227</td><td>49</td><td>1</td></tr>
</table>

Half-size character code list

Character	Code	Character	Code	Character	Code	Character	Code	Character	Code	Character	Code	Character	Code	Character	Code
A	65	Q	81	6	54	,	44	ｲ	-78	ﾂ	-62	ﾒ	-46	ｪ	-86
B	66	R	82	7	55	-	45	ｳ	-77	ﾃ	-61	ﾓ	-45	ｫ	-85
C	67	S	83	8	56	.	46	ｴ	-76	ﾄ	-60	ﾔ	-44	ｬ	-84
D	68	T	84	9	57	/	47	ｵ	-75	ﾅ	-59	ﾕ	-43	ｭ	-83
E	69	U	85		32	:	58	ｶ	-74	ﾆ	-58	ﾖ	-42	ｮ	-82
F	70	V	86	!	33	;	59	ｷ	-73	ﾇ	-57	ﾗ	-41	ｯ	-81
G	71	W	87	"	34	<	60	ｸ	-72	ﾈ	-56	ﾘ	-40	ﾞ	-34
H	72	X	88	#	35	=	61	ｹ	-71	ﾉ	-55	ﾙ	-39	ﾟ	-33
I	73	Y	89	$	36	>	62	ｺ	-70	ﾊ	-54	ﾚ	-38	｡	-95
J	74	Z	90	%	37	?	63	ｻ	-69	ﾋ	-53	ﾛ	-37	｢	-94
K	75	0	48	&	38	@	64	ｼ	-68	ﾌ	-52	ﾜ	-36	｣	-93
L	76	1	49	'	39	[	91	ｽ	-67	ﾍ	-51	ｦ	-90	､	-92
M	77	2	50	(	40	¥	92	ｾ	-66	ﾎ	-50	ﾝ	-35	･	-91
N	78	3	51	)	41	]	93	ｿ	-65	ﾏ	-49	ｧ	-89		
O	79	4	52	*	42	_	95	ﾀ	-64	ﾐ	-48	ｨ	-88		
P	80	5	53	+	43	ｱ	-79	ﾁ	-63	ﾑ	-47	ｩ	-87		

NOTE

The dakuten (ﾞ)(voiced sound mark), the han-dakuten(ﾟ)(semi-voiced sound mark) in Japanese katakana are counted as one half-size character.

Full-size character code list (a)

Chara cter	Code High	Code Low	Chara cter	Code High	Code Low	Chara cter	Code High	Code Low	Chara cter	Code High	Code Low
ぁ	2	0	ち	2	64	め	2	-128	面	2	-64
あ	2	2	ぢ	2	66	も	2	-126	最	2	-62
ぃ	2	4	っ	2	68	ゃ	2	-124	小	2	-60
い	2	6	つ	2	70	や	2	-122	内	2	-58
ぅ	2	8	づ	2	72	ゅ	2	-120	大	2	-56
う	2	10	て	2	74	ゆ	2	-118	加	2	-54
ぇ	2	12	で	2	76	ょ	2	-116	工	2	-52
え	2	14	と	2	78	よ	2	-114	切	2	-50
ぉ	2	16	ど	2	80	ら	2	-112	削	2	-48
お	2	18	な	2	82	り	2	-110	倣	2	-46
か	2	20	に	2	84	る	2	-108	正	2	-44
が	2	22	ぬ	2	86	れ	2	-106	途	2	-42
き	2	24	ね	2	88	ろ	2	-104	中	2	-40
ぎ	2	26	の	2	90	ゎ	2	-102	荒	2	-38
く	2	28	は	2	92	わ	2	-100	具	2	-36
ぐ	2	30	ば	2	94	素	2	-98	番	2	-34
け	2	32	ぱ	2	96	材	2	-96	号	2	-32
げ	2	34	ひ	2	98	を	2	-94	仕	2	-30
こ	2	36	び	2	100	ん	2	-92	上	2	-28
ご	2	38	ぴ	2	102	種	2	-90	込	2	-26
さ	2	40	ふ	2	104	類	2	-88	点	2	-24
ざ	2	42	ぶ	2	106	棒	2	-86	方	2	-22
し	2	44	ぷ	2	108	穴	2	-84	向	2	-20
じ	2	46	へ	2	110	成	2	-82	速	2	-18
す	2	48	べ	2	112	形	2	-80	度	2	-16
ず	2	50	ぺ	2	114	質	2	-78	送	2	-14
せ	2	52	ほ	2	116	寸	2	-76	量	2	-12
ぜ	2	54	ぼ	2	118	法	2	-74	開	2	-10
そ	2	56	ぽ	2	120	外	2	-72	始	2	-8
ぞ	2	58	ま	2	122	径	2	-70	深	2	-6
た	2	60	み	2	124	長	2	-68	主	2	-4
だ	2	62	む	2	126	端	2	-66	軸	2	-2

Full-size character code list (b)

Character	Code		Character	Code		Character	Code		Character	Code	
	High	Low		High	Low		High	Low		High	Low
回	3	0	源	3	64	設	3	-128	無	3	-64
転	3	2	投	3	66	定	3	-126	視	3	-62
数	3	4	入	3	68	一	3	-124	器	3	-60
位	3	6	間	3	70	覧	3	-122	原	3	-58
置	3	8	分	3	72	表	3	-120	登	3	-56
決	3	10	秒	3	74	部	3	-118	録	3	-54
直	3	12	自	3	76	炭	3	-116	再	3	-52
線	3	14	運	3	78	合	3	-114	処	3	-50
時	3	16	負	3	80	金	3	-112	理	3	-48
円	3	18	荷	3	82	鋼	3	-110	描	3	-46
反	3	20	実	3	84	超	3	-108	画	3	-44
現	3	22	使	3	86	硬	3	-106	過	3	-42
在	3	24	用	3	88	先	3	-104	容	3	-40
指	3	26	寿	3	90	付	3	-102	編	3	-38
令	3	28	命	3	92	摩	3	-100	集	3	-36
値	3	30	新	3	94	耗	3	-98	未	3	-34
領	3	32	規	3	96	仮	3	-96	対	3	-32
域	3	34	除	3	98	想	3	-94	相	3	-30
診	3	36	隅	3	100	副	3	-92	座	3	-28
断	3	38	取	3	102	行	3	-90	標	3	-26
操	3	40	単	3	104	挿	3	-88	示	3	-24
作	3	42	補	3	106	消	3	-86	名	3	-22
手	3	44	能	3	108	去	3	-84	歯	3	-20
引	3	46	独	3	110	山	3	-82	変	3	-18
機	3	48	終	3	112	高	3	-80	呼	3	-16
械	3	50	了	3	114	準	3	-78	推	3	-14
残	3	52	記	3	116	備	3	-76	馬	3	-12
移	3	54	角	3	118	完	3	-74	力	3	-10
動	3	56	溝	3	120	後	3	-72	系	3	-8
次	3	58	刃	3	122	弧	3	-70	選	3	-6
早	3	60	幅	3	124	助	3	-68	達	3	-4
電	3	62	広	3	126	択	3	-66	閉	3	-2

Full-size character code list (c)

Chara cter	Code		Chara cter	Code		Chara cter	Code		Chara cter	Code	
	High	Low		High	Low		High	Low		High	Low
禁	4	0	範	4	64	絵	4	-128	控	4	-64
復	4	2	囲	4	66	階	4	-126	更	4	-62
帰	4	4	倍	4	68	概	4	-124	校	4	-60
書	4	6	率	4	70	該	4	-122	構	4	-58
個	4	8	注	4	72	巻	4	-120	根	4	-56
桁	4	10	側	4	74	換	4	-118	左	4	-54
稼	4	12	特	4	76	気	4	-116	差	4	-52
由	4	14	殊	4	78	起	4	-114	雑	4	-50
両	4	16	距	4	80	軌	4	-112	参	4	-48
半	4	18	離	4	82	技	4	-110	散	4	-46
逃	4	20	連	4	84	疑	4	-108	産	4	-44
底	4	22	続	4	86	供	4	-106	算	4	-42
逆	4	24	増	4	88	共	4	-104	治	4	-40
下	4	26	隔	4	90	境	4	-102	耳	4	-38
空	4	28	件	4	92	強	4	-100	式	4	-36
四	4	30	初	4	94	教	4	-98	失	4	-34
触	4	32	期	4	96	掘	4	-96	修	4	-32
平	4	34	条	4	98	繰	4	-94	十	4	-30
代	4	36	経	4	100	係	4	-92	従	4	-28
辺	4	38	握	4	102	傾	4	-90	勝	4	-26
格	4	40	圧	4	104	型	4	-88	商	4	-24
子	4	42	扱	4	106	検	4	-86	少	4	-22
周	4	44	陰	4	108	権	4	-84	尚	4	-20
心	4	46	隠	4	110	研	4	-82	昇	4	-18
本	4	48	右	4	112	肩	4	-80	植	4	-16
群	4	50	押	4	114	見	4	-78	色	4	-14
停	4	52	横	4	116	験	4	-76	食	4	-12
止	4	54	黄	4	118	元	4	-74	伸	4	-10
巾	4	56	億	4	120	弦	4	-72	信	4	-8
微	4	58	屋	4	122	減	4	-70	侵	4	-6
状	4	60	化	4	124	孔	4	-68	振	4	-4
路	4	62	何	4	126	巧	4	-66	浸	4	-2

Full-size character code list (d)

Character	Code		Character	Code		Character	Code		Character	Code	
	High	Low		High	Low		High	Low		High	Low
真	5	0	継	5	64	図	5	-128	口	5	-64
暗	5	2	計	5	66	違	5	-126	語	5	-62
以	5	4	軽	5	68	印	5	-124	誤	5	-60
意	5	6	言	5	70	沿	5	-122	交	5	-58
異	5	8	限	5	72	遠	5	-120	厚	5	-56
影	5	10	互	5	74	央	5	-118	項	5	-54
鋭	5	12	降	5	76	奥	5	-116	刻	5	-52
越	5	14	採	5	78	往	5	-114	告	5	-50
価	5	16	済	5	80	応	5	-112	黒	5	-48
可	5	18	細	5	82	会	5	-110	財	5	-46
科	5	20	姿	5	84	解	5	-108	策	5	-44
果	5	22	思	5	86	改	5	-106	系	5	-42
箇	5	24	写	5	88	割	5	-104	試	5	-40
課	5	26	射	5	90	活	5	-102	資	5	-38
各	5	28	斜	5	92	願	5	-100	事	5	-36
拡	5	30	者	5	94	基	5	-98	持	5	-34
核	5	32	車	5	96	奇	5	-96	似	5	-32
学	5	34	借	5	98	寄	5	-94	釈	5	-30
掛	5	36	縦	5	100	岐	5	-92	弱	5	-28
漢	5	38	重	5	102	既	5	-90	受	5	-26
簡	5	40	出	5	104	近	5	-88	収	5	-24
観	5	42	述	5	106	区	5	-86	純	5	-22
関	5	44	術	5	108	矩	5	-84	順	5	-20
含	5	46	渉	5	110	駆	5	-82	所	5	-18
却	5	48	照	5	112	偶	5	-80	序	5	-16
客	5	50	省	5	114	旧	5	-78	剰	5	-14
休	5	52	章	5	116	求	5	-76	場	5	-12
急	5	54	証	5	118	球	5	-74	常	5	-10
業	5	56	象	5	120	究	5	-72	飾	5	-8
曲	5	58	身	5	122	級	5	-70	水	5	-6
均	5	60	進	5	124	欠	5	-68	錐	5	-4
筋	5	62	人	5	126	結	5	-66	据	5	-2

Full-size character code list (e)

Character	Code		Character	Code		Character	Code		Character	Code	
	High	Low		High	Low		High	Low		High	Low
制	6	0	必	6	64	替	6	-128	木	6	-64
整	6	2	百	6	66	段	6	-126	目	6	-62
製	6	4	複	6	68	知	6	-124	歪	6	-60
前	6	6	物	6	70	地	6	-122	揺	6	-58
全	6	8	文	6	72	致	6	-120	様	6	-56
然	6	10	聞	6	74	遅	6	-118	溶	6	-54
則	6	12	併	6	76	追	6	-116	要	6	-52
属	6	14	忘	6	78	通	6	-114	抑	6	-50
即	6	16	末	6	80	伝	6	-112	良	6	-48
他	6	18	密	6	82	得	6	-110	輪	6	-46
多	6	20	有	6	84	読	6	-108	和	6	-44
存	6	22	余	6	86	凸	6	-106	話	6	-42
谷	6	24	与	6	88	凹	6	-104	枠	6	-40
探	6	26	裏	6	90	突	6	-102	節	6	-38
短	6	28	立	6	92	鈍	6	-100	説	6	-36
徴	6	30	略	6	94	敗	6	-98	絶	6	-34
鎖	6	32	青	6	96	杯	6	-96	千	6	-32
調	6	34	席	6	98	背	6	-94	専	6	-30
頂	6	36	石	6	100	配	6	-92	浅	6	-28
鉄	6	38	積	6	102	品	6	-90	旋	6	-26
添	6	40	赤	6	104	不	6	-88	総	6	-24
頭	6	42	接	6	106	布	6	-86	走	6	-22
同	6	44	折	6	108	並	6	-84	退	6	-20
導	6	46	粗	6	110	頁	6	-82	台	6	-18
道	6	48	創	6	112	別	6	-80	第	6	-16
熱	6	50	双	6	114	片	6	-78	題	6	-14
年	6	52	捜	6	116	返	6	-76	卓	6	-12
濃	6	54	太	6	118	勉	6	-74	室	6	-10
箱	6	56	打	6	120	弁	6	-72	着	6	-8
発	6	58	体	6	122	保	6	-70	柱	6	-6
抜	6	60	待	6	124	明	6	-68	鋳	6	-4
伴	6	62	態	6	126	滅	6	-66	丁	6	-2

Full-size character code list (f)

Character	Code		Character	Code		Character	Code		Character	Code	
	High	Low		High	Low		High	Low		High	Low
低	7	0	戻	7	64		7	-128	界	7	-64
訂	7	2	冷	7	66		7	-126	混	7	-62
肉	7	4	垂	7	68	貫	7	-124	監	7	-60
日	7	6	緑	7	70	安	7	-122	締	7	-58
白	7	8	紫	7	72	α	7	-120	護	7	-56
薄	7	10	許	7	74	β	7	-118	己	7	-54
比	7	12	測	7	76	程	7	-116	称	7	-52
皮	7	14	精	7	78	抗	7	-114	樹	7	-50
被	7	16	効	7	80	張	7	-112	脂	7	-48
非	7	18	→	7	82	任	7	-110	料	7	-46
美	7	20		7	84	破	7	-108	落	7	-44
普	7	22	↑	7	86	損	7	-106	確	7	-42
伏	7	24		7	88	御	7	-104	認	7	-40
歩	7	26	←	7	90	足	7	-102	報	7	-38
包	7	28		7	92	守	7	-100	排	7	-36
門	7	30	↓	7	94	般	7	-98	性	7	-34
問	7	32		7	96	納	7	-96	生	7	-32
縮	7	34		7	98	義	7	-94	績	7	-30
列	7	36		7	100	丸	7	-92	判	7	-28
万	7	38		7	102	汎	7	-90	搬	7	-26
利	7	40		7	104	固	7	-88	砥	7	-24
訳	7	42		7	106	毎	7	-86	θ	7	-22
礼	7	44	板	7	108	当	7	-84	島	7	-20
乱	7	46	予	7	110	的	7	-82	壁	7	-18
放	7	48		7	112	詳	7	-80		7	-16
枚	7	50	家	7	114	鳥	7	-78		7	-14
約	7	52	装	7	116	適	7	-76		7	-12
練	7	54	管	7	118	論	7	-74		7	-10
油	7	56	粉	7	120	額	7	-72		7	-8
劣	7	58	等	7	122	縁	7	-70	//	7	-6
例	7	60		7	124	温	7	-68		7	-4
郭	7	62		7	126	給	7	-66		7	-2

NOTE

1 The character can be displayed up to the character code just before the parameter Nos. 7220 to 7399 is set to 0, in the name of a general-purpose switch on the software operator's panel. Therefore, if using the space, specify the character code “32” (the space of the half-size character).

Ex1) When the parameter is set to 0 in the half-size character

Parameter	Setting value	Character
7220	3	回
7221	0	
7222	3	転
7223	2	
7224	0	
7225	49	1
7226	50	2
7227	51	3

The character string of display : “回転”

Ex2) When the parameter is set to 32 (the space of the half-size character) in the half-size character

Parameter	Setting value	Character
7220	3	回
7221	0	
7222	3	転
7223	2	
7224	32	(space)
7225	49	1
7226	50	2
7227	51	3

The character string of display : “回転 123”

2 Do not specify the character codes of the blank in the “Full-size character code list”. When it is specified, the space of the full-size character or an invalid character is displayed.

3 Please specify two spaces of the half-size character to display a space of the full-size character.

Note

NOTE

1 Only the modes shown below can be selected by soft switches. When the mode for DNC operation is to be required, then, all control switches for mode selection should be on the machine operator's panel or a general-purpose soft switch should be used to select the mode for DNC operation.
<u>Soft switches available for mode selection</u>
- Manual data input
- Automatic operation
- Memory edit
- Manual handle feed / incremental feed
- Jog feed
- Manual reference position return

2 Only one soft switch is available for the protection key. But, four input signals are available for protection key (KEY1, KEY2, KEY3 and KEY4).
Generally, four input signals are simultaneously turned to 1 or 0 according to the state of the protection soft switch.
For a multi-path system, the signals are as follows. These signals are common to all paths.
- KEYO<F0075.6>
- KEY1 to KEY4<G0046.3 to G0046.6>

3 When the soft switch for feed hold is turned on, output signal SPO is turned to 1, and the PMC turns feed hold signal *SP to 0.
In contrast to the above, when the soft switch for feed hold is turned off, output signal SPO is turned 0 and the PMC turns signal *SP to 1. For soft switches other than feed hold and general soft switches, when an output signal corresponding to a soft switch is turned to 1, the corresponding input signal is turned to 1.

The Table 13.1.2 (a) lists the jog feedrate override values which can be selected by soft switches.

Table 13.1.2 (a)

Override values (%)	*JV0O to *JV15O			
	15	8	7	0
0	1111	1111	1111	1111
0.1	1111	1111	1111	0101
0.14	1111	1111	1111	0001
0.2	1111	1111	1110	1011
0.27	1111	1111	1110	0100
0.37	1111	1111	1101	1010
0.52	1111	1111	1100	1011
0.72	1111	1111	1011	0111
1.0	1111	1111	1001	1011
1.4	1111	1111	0111	0011
2.0	1111	1111	0011	0111
2.7	1111	1110	1111	0001
3.7	1111	1110	1000	1101
5.2	1111	1101	1111	0111
7.2	1111	1101	0010	1111
10.0	1111	1100	0001	0111
14.0	1111	1010	1000	0111
20.0	1111	1000	0010	1111
27.0	1111	0101	0111	0011
37.0	1111	0001	1000	1011
52.0	1110	1011	1010	1111
72.0	1110	0011	1101	1111
100.0	1101	1000	1110	1111
140.0	1100	1001	0100	1111
200.0	1011	0001	1101	1111

The Table 13.1.2 (b) lists the feedrate override values which can be selected by soft switches.

Table 13.1.2 (b)

Override values (%)	*FV0O to *FV7O	
	7	0
0	1111	1111
10	1111	0101
20	1110	1011
30	1110	0001
40	1101	0111
50	1100	1101
60	1100	0011
70	1011	1001
80	1010	1111
90	1010	0101
100	1001	1011
110	1001	0001
120	1000	0111
130	0111	1101
140	0111	0011
150	0110	1001
160	0101	1111
170	0101	0101
180	0100	1011
190	0100	0001
200	0011	0111

NOTE
The software operator's panel general-purpose switch function enables the use of general-purpose switches 1 to 16.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Display and setting of the software operator's panel

13.1.3 8-Level Data Protection Function

Overview

Eight operation levels can be set for CNC and PMC operation and eight protection levels can be set for various types of CNC and PMC data.
When various types of CNC and PMC data are changed or output externally, the system compares the operation level with the protection level to determine whether change or external output is allowed.

Explanation

- Operation level

Eight operation levels can be set for CNC and PMC operation.
Operation levels 0 to 3 are selected by the memory protection key signal.
Operation levels 4 to 7 are selected by a password.

Operation level	Setting method	Sample classification
7 (High)	Password	-
6	Password	MTB
5	Password	Dealer, Integrator
4	Password	End user
3	Memory protection key signal	User level (Level 1)
2	Memory protection key signal	User level (Level 2)
1	Memory protection key signal	User level (Level 3)
0 (low)	Memory protection key signal	User level (Level 4)

NOTE
1 Once any of operation levels 4 to 7 is set, it is retained (even when the power is turned off) until clear operation is performed by the corresponding password.
2 The initial values of the passwords for operation levels 4 to 6 are shown below.
- Operation level 4: "HC3V9ZEP"
- Operation level 5: "J72WB8YA"
- Operation level 6: "VLR6T92M"
3 Operation level 7 is reserved for the maintenance of the CNC and PMC.

⚠ **CAUTION**
When this function is provided, the conventional memory protection function is disabled.
When this function is added, the programmer protection function of the PMC is disabled. However, the sequence program password function can be used in combination with this function.

- Data protection level

A data protection level can be set for each of the following types of data. There are two data protection levels as shown below.

- Change protection level
  Protection level used for changing data
- Output protection level
  Protection level used for externally outputting data.

Protection levels 0 (low) to 7 (high) can be set.

CNC data protection level setting items

Data type	Initial value of protection level	
	Change	Output
Custom macro conversion data (including variable data specific to the macro executor)	0	0
Periodical maintenance information data	0	0
Tool management data	0	0
Tool offset data (for each type when classification is performed by figure and figure/wear)	0	0
Clock data	0	0
Workpiece origin shift data	0	0
Workpiece origin offset data	0	0
Ethernet setting data	0	0
Parameter data	4	0
Setting data	0	0
Power Mate CNC Manager function parameter data	0	0
Each part program	0	0
Part program editing	0	0
Presetting an absolute coordinate	0	0

PMC data protection level setting items

Data type	Initial value of protection level	
	Change	Output
Configuration parameter	0	0
Setting (online)	0	0
Setting (for each path)	0	0
Sequence parameter	0	0
PMC parameter	0	0
Timer	0	0
Counter	0	0
Keep relay	0	0
Keep relay (system)	0	0
Data table	0	0
Data table control	0	0
PMC memory	0	0
I/O configuration	0	0
I/O Link group selection	0	0
Registration of I/O device	0	0

- Changing or externally outputting of data

When various types of CNC and PMC data is changed or output externally, the change protection level or output protection level set for the target data is compared with its current operation level.
If the operation level is equal to or higher than the protection level set for the target data (operation level ≥ protection level), it is assumed that the target data can be changed or output externally and the data is allowed to be changed or output externally.
The operation level must be changed according to the purpose as needed.
The protection level must be set according to the confidentiality and severity of data.

[Example of application]

(1) Set the change protection level as follows.
- Parameter (Change protection level 4)
- Tool offset data (Change protection level 0)

(2) The changeable data can be restricted by changing the operation level during CNC and PMC operation.
- Operation level 4
  Parameters and tool offset data can be changed.
- Operation level 0
  Tool offset data can be changed.
  Parameters cannot be changed.

Data / Operation level	Parameter	Tool offset data
4	Changeable	Changeable
0	Not changeable	Changeable

Notes

- Tool management data

NOTE
Data related to tool information on the individual tool data screen of the tool management function is protected on the tool offset data protection level.

- Tool offset data

NOTE
1 Valid types of tool offset data vary depending on the tool offset memory used. See the tables below when setting a data protection level.
2 When a type of data that is not permitted to be changed or externally output is found during tool offset data input/output, the following operation takes place.
- Input : Data of other than the data type that is not permitted to be changed is changed.
- Output : Data of other than the data type that is not permitted to be changed is output.

M

Data type/tool offset memory	Tool offset memory A	Tool offset memory B	Tool offset memory B
Tool offset data	Applicable	Not applicable	Not applicable
Tool offset data (geometry)	Not applicable	Applicable	Not applicable
Tool offset data (wear)	Not applicable	Applicable	Not applicable
Tool offset data (tool radius, geometry)	Not applicable	Not applicable	Applicable
Tool offset data (tool radius, wear)	Not applicable	Not applicable	Applicable
Tool offset data (tool length, geometry)	Not applicable	Not applicable	Applicable
Tool offset data (tool length, wear)	Not applicable	Not applicable	Applicable

Data type/tool offset memory	Virtual tool tip direction
Tool offset data	Applicable

T

Data type/tool offset memory	Without tool geometry and wear compensation	With tool geometry and wear compensation
Tool offset data	Applicable	Not applicable
Tool offset data (geometry)	Not applicable	Applicable
Tool offset data (wear)	Not applicable	Applicable

Data type/tool offset memory	Tool radius compensation Y-axis offset (without tool geometry and wear compensation)	Tool radius compensation Y-axis offset (with tool geometry and wear compensation)
Tool offset data	Applicable	Not applicable
Tool offset data (geometry)	Not applicable	Applicable
Tool offset data (wear)	Not applicable	Applicable

Data type/tool offset memory	Virtual tool tip direction
Tool offset data	Applicable

Data type/tool offset memory	Second geometry tool offset data
Tool offset data (geometry)	Applicable

- Part programs and part program editing

NOTE

1. When changing the protection level of a part program, use the program list screen instead of the protection level setting screen. See the description of "Operation/setting screen".
2. Part program editing involves editing of programs for the MDI mode.

- Absolute coordinate preset operation

NOTE

When absolute coordinates are preset, workpiece coordinate system presetting is protected.

- Other notes

NOTE

1. For some data, the output function is not provided.
2. When a higher protection level than the current operation level is set for data, that protection level cannot be changed.
3. The protection level of data cannot be changed to a protection level higher than the current operation level.
4. Part program editing involves editing of programs for the MDI mode.
5. For details of the protection level of PMC data, refer to "PMC Programming Manual (B-64513EN)".
6. In principle, the data change protection check is performed for changes made by MDI. Changes made by machine operations and so on are not checked.
   For example, a programmable parameter input by specifying G10L50 may be changed regardless of the operation level and parameter change protection level.
   To protect data from illegal programmed commands, take appropriate measures; for example, set an appropriate change protection level for program edit operations not to create illegal programs.

Signal

Memory protection signals KEY1 to KEY4<G0046.3 to G0046.6>

[Classification] Input signal

[Function] These signals select the operation level for the 8-level data protection function.
The correspondence between signals and operation levels is shown below.

Operation level	KEY4	KEY3	KEY2	KEY1
3	0	1	0	0
2	0	0	1	0
1	0	0	0	1
0	0	0	0	0

NOTE
When a combination other than the above is set, operation level 0 is assumed.

[Caution] When the 8-level data protection function is not used, these signals are used as memory protection keys.
Note that what these signals indicate changes depending on whether the 8-level data protection function is used.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0046		KEY4	KEY3	KEY2	KEY1			

Diagnosis data

1004	Current level of 8-Level Data Protection Function

[Data type] Word

[Unit of data] None

[Valid data range] 0 to 7

Current operation level of 8-Level Data Protection Function

Operation/setting screen

Various settings or display about operation levels or protection levels can be performed on the following screens.

- Password change screen
- Operation level setting screen
- Protection level setting screen
- Program list screen

- Password change screen

On the password change screen, the following display or operations can be performed.

(1) Displaying the current operation level

(2) Changing the passwords of operation levels 4 to 7

NOTE
1 A password consists of 3 to 8 characters including the following.
- Uppercase letter
- Numeric

NOTE

2 When a password is entered, * is displayed instead of each of input characters.
3 The following shows whether passwords can be changed at the current operation level.
  - Password having a higher operation level than the current operation level
    Cannot be changed.
  - Password having the same operation level as the current operation level
    Can be changed.
  - Password having a lower operation level than the current operation level
    Can be changed (reverting to the initial password can only be performed).

⚠ CAUTION

The set password is not displayed.
Be careful not to forget the password.

- Operation level setting screen

On the operation level setting screen, the following display or operations can be performed.

(1) Displaying the current operation level
(2) Entering the password and then selecting one of operation levels 4 to 7
(3) Canceling the entered password and then selecting the operation level other than operation levels 4 to 7

NOTE

When a password is entered, * is displayed instead of each character.

- Protection level setting screen

On the protection level setting screen, the following display or operations can be performed.

(1) Displaying the current operation level
(2) Displaying the change protection level and output protection level of each data
(3) Changing the change protection level and output protection level of each data

NOTE

1 For data whose protection level is higher than the operation level, the protection level cannot be changed.
2 The protection level cannot be changed to a protection level that is higher than the current operation level.

- Program list screen

On the program list screen, the following display and operations can be performed.

(1) Displaying the change protection level and output protection level of each part program
(2) Changing the change protection level and output protection level of each part program

NOTE

1 For data whose protection level is higher than the operation level, the protection level cannot be changed.
2 The protection level cannot be changed to a protection level that is higher than the current operation level.

13.1.4 Touch Panel Control

⚠ **WARNING**

FANUC's touch panel is an analog resistive film type. When two or more points are pressed at the same time, there is a possibility that it behaves as if the center of these points was pressed, and this wrong output or malfunction may cause an accident. Do not create a virtual machine operator's panel screens on which two or more points are pressed at the same time for touch panel operation.

On the virtual machine operator panel screen, never support safety-related operations that may lose human life or may cause serious damage, or real-time operations such as emergency stop, program start, program stop, axis movements, etc. If there is a failure in the CNC, peripheral units, or cable, wrong outputs or malfunctions may cause an accident. In addition, real-time operation is not guaranteed on the touch panel screen.

Overview

A display unit with a touch panel enables you to operate soft keys by touching the screen.

Moreover, an application using a touch panel can be created with the C Language Executor.

NOTE

1. With a CNC of LCD-mounted type, RS-232C serial port 2 (JD36A) is occupied.
2. With a CNC of stand-alone type, a serial port dedicated to a touch panel is used.
3. Touch panel pressing information is read at intervals of 32 msec.
4. A positional precision of ±2.5 mm is provided.

Explanation

- C Language Executor

With the C Language Executor, touch panel functions can be used. For the specifications of the functions, refer to "C Language Executor Operator's Manual".

- Calibration

When replacing the touch panel or clearing all memory (SRAM) data, set data for calibrating the positional relation between the touch panel and LCD according to the procedure below.

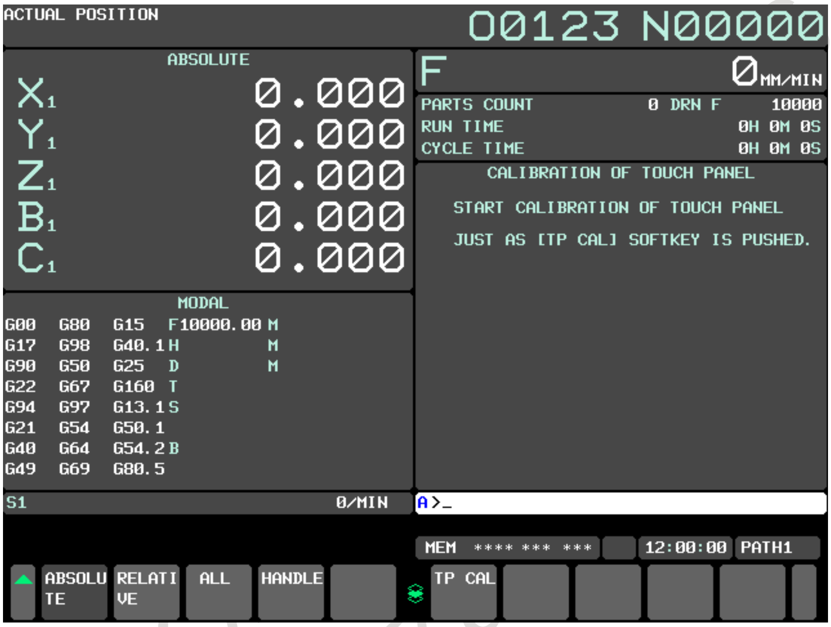
Calibration procedure

1 Enable the touch panel calibration screen.(Set bit 5 (DCL) of parameter No. 3113 to 1.)

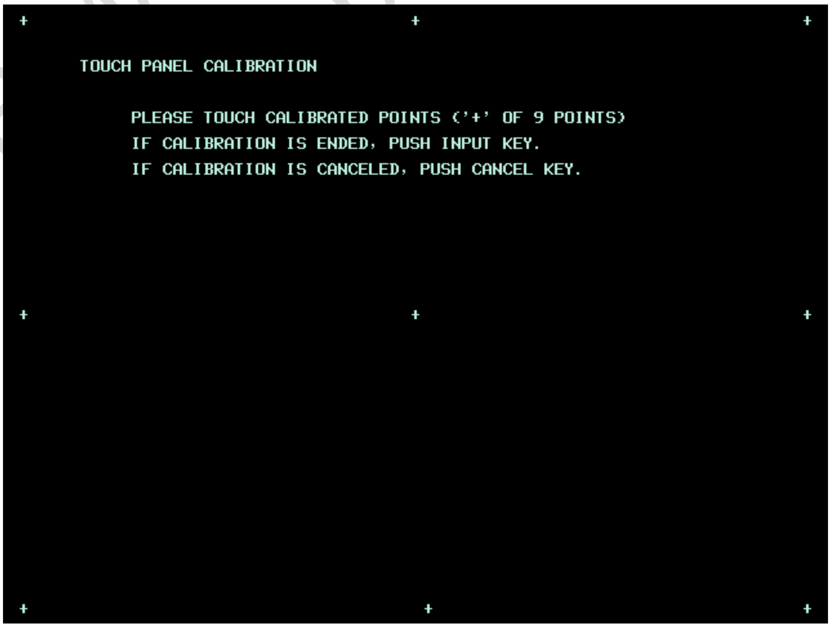
2 Press function key SYSTEM.

3 Press the continuous menu key several times. The [TOUCH PANEL] soft key is displayed.

4 Press the [TOUCH PANEL] soft key then the [(OPRT)] soft key. The [TP CAL] soft key is displayed.



5 Pressing the [TP CAL] soft key displays the touch panel calibration screen for all screens.



6 Press the calibration points (9 points) with a dedicated pen. When a point is pressed correctly, the "+" mark changes to the "O" mark. If a "+" mark is not pressed accurately, the message "CALIBRATION POINT DOES NOT MATCH. PLEASE PUSH AGAIN." is displayed.

7 After entering the calibration points (9 points), press INPUT key to complete calibration. To cancel calibration or retry, press CAN key. The screen display returns to the previous screen. If INPUT key is pressed before entering the calibration points (9 points), calibration operation is cancelled.

8 When calibration is terminated normally, the message "CALIBRATION WAS ENDED." is displayed.

9 Upon completion of calibration, disable the touch panel calibration screen to protect against operation errors. (Set bit 5 (DCL) of parameter No. 3113 to 0.)

NOTE
1 If the screen display is switched from the touch panel calibration screen to another screen such as the alarm screen by the automatic screen switching function, calibration is automatically cancelled.
2 Calibration operation can be performed in any mode.
3 After system startup, perform calibration operation swiftly before starting operation.

- Operation when two points are pressed

When the touch panel is pressed at two or more points, the position of the gravity center is obtained by considering how each of these points is pressed, and the touch panel is assumed to be pressed at this gravity center position. At this time the coordinates that can be obtained can be set by setting bit 1 (T2P) of parameter No. 3192.

NOTE
When two points are pressed in a row in a short time interval, the touch panel may react as if the center of the two points was pressed, even though parameter T2P (No. 3192#1) is set to 1.

(1) Suppose that in Fig. 13.1.4 (a), as soon as point A is pressed, point B is pressed.

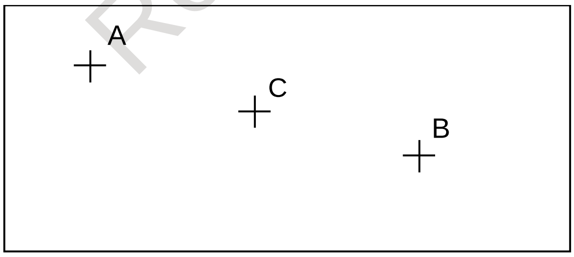


Fig. 13.1.4 (a) Pressed points on the touch panel

(2) The coordinates that can be obtained change as shown in Fig. 13.1.4 (b).

On	Cont'd	. . .	Cont'd	Cont'd	Cont'd	. . .	Off
A	A		A	C	C		

Fig. 13.1.4 (b) Coordinates when parameter T2P = 0

(3) If bit T2P is set to 1, A is assumed to be held pressed even after the pressed point changes from point A to point C (Fig. 13.1.4 (c)).

On	Cont'd	. . .	Cont'd	Cont'd	Cont'd	. . .	Off
A	A		A	A	A		

Fig. 13.1.4 (c) Coordinates when parameter T2P = 1

- Operation when dragging is performed

When dragging (continuously pressing the touch panel while making a movement) is performed on the touch panel, the system response varies depending on the setting of bit 1 (T2P) of parameter No. 3192. When the drag function is used in C Language Executor applications, set T2P to 0.

1) Suppose that in Fig. 13.1.4 (d), point A is pressed first and then dragging is performed from point A to B to C.

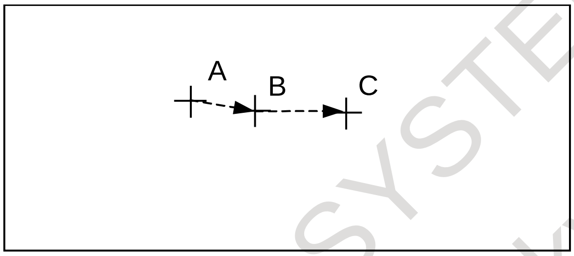


Fig. 13.1.4 (d) Dragging on the touch panel

2) The coordinates that can be obtained change as shown in Fig. 13.1.4 (e).

On	Cont'd	Cont'd
A	B	C

Fig. 13.1.4 (e) Coordinates when parameter T2P = 0

3) If bit T2P is set to 1, A is assumed to be kept pressed even after the pressed point moves from point A to point C (Fig. 13.1.4 (f)).

On	Cont'd	Cont'd
A	A	A

Fig. 13.1.4 (f) Coordinates when parameter T2P = 1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3113			DCL					

[Input type] Parameter input
[Data type] Bit

#5 DCL The touch panel compensation screen is:
0: Disabled.
1: Enabled.
Set this parameter to 0 usually. Touch panel compensation becomes necessary only when the panel is replaced or memory all clear operation is performed. Set this parameter to 1 only when performing touch panel compensation. Upon completion of compensation, set this parameter to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
3119						DDS		

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#2 DDS The touch panel is:
0: Enabled.
1: Disabled.
Set this parameter to 1 when disabling the touch panel temporarily, for example, at start-up time.

	#7	#6	#5	#4	#3	#2	#1	#0
3192						TRA	T2P	

[Input type] Parameter input
[Data type] Bit

#1 T2P When more than one point is pressed on the touch panel:
0: The position at the center of gravity is obtained.
1: The point pressed first is obtained.

> **NOTE**
> 1 When two points are pressed in a row in a short time interval, the touch panel may react as if the center of the two points was pressed, even though parameter T2P (No. 3192#1) is set to 1.
> 2 If a C executer application or the like has a touch panel drag (move in pressed state) function, set parameter T2P to 0.

#2 TRA If a point on the touch panel is kept pressed for a time specified in parameter No. 3197 or longer,
0: An alarm is not raised.
1: An alarm SR5303, “TOUCH PANEL ERROR” is raised.

> **NOTE**
> 1 If an C executer application or the like has a touch panel repeat (continue pressing) function, set parameter TRA to 0.
> 2 In personal computer functions, the parameter is valid just for the CNC screen display function.

3197	Detection time of continuous pressing on touch panel

[Input type] Parameter input
[Data type] Word
[Unit of data] sec
[Valid data range] 0 to 255
Set a period of continuous pressing on the touch panel which causes alarm SR5303, “TOUCH PANEL ERROR” to be raised. When 0 is set, it is equivalent to 20.

> **NOTE**
> This parameter is valid when bit 2 (TRA) of parameter No. 3192 is set to 1.

Alarm and message

Number	Message	Description
SR5303	TOUCH PANEL ERROR	The touch panel is not connected correctly, or the touch panel cannot be initialized when the power is turned on. If bit 2 (TRA) of parameter No. 3192 is set to 1, this message is issued also when the touch panel is being kept pressed. Correct the cause then turn on the power again.

Caution

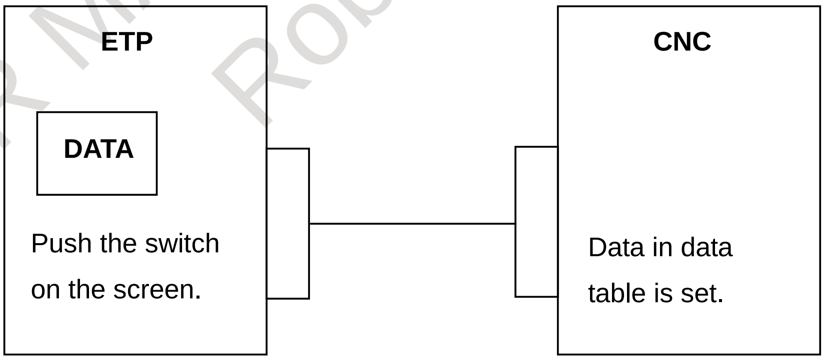
> ⚠ **CAUTION**
> When all memory (SRAM) data is cleared, the soft keys on the touch panel are not made usable yet. In this case, the MDI keys (such as the cursor keys and page keys) need to be used for setting.

13.1.5 External Touch Panel Interface

Outline

External Touch Panel (called "ETP" below) of SNP-X protocol can be connected with Series 30*i*.
ETP has functions that can read out/ write in from/to PMC such control signals as input signal(X), output signal(Y), internal relay(R), keep relay(K), data table(D), extra relay(E), timer(T), counter(C), and the function is almost the same as operating panel of machine.

The remarkable function of ETP is drawing function.
Assignment between drawing and address (signal) can be specified freely.
For example, the data in data table can be set with the switch on the screen which is designed to assign the setting of data table.

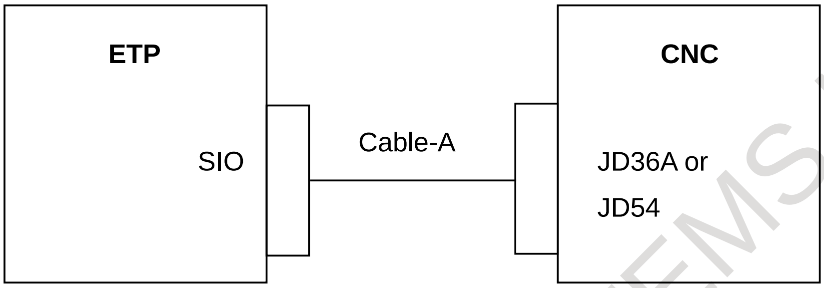


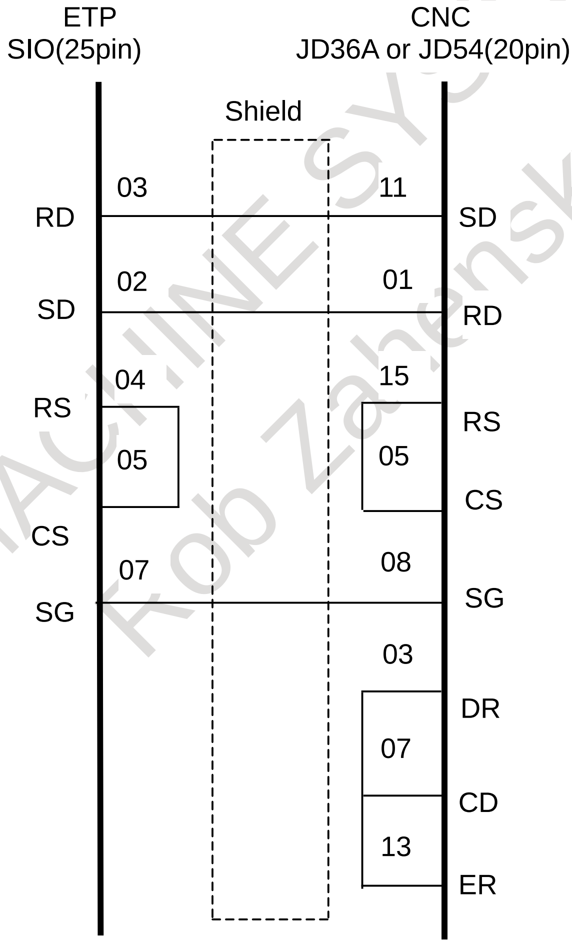
Explanation

- Connection

RS-232C serial port 2(JD36A; for the 15”/19” LCD-mounted type CNC and LCD-mounted type CNC with the is functions, JD54) on main CPU board is used in CNC.
Cable-A must be based on RS-232C standard, and are prepared by customer.

(1) Cable-A (connection diagram between JD36A or JD54 on CNC and SIO on ETP.)





Cables must be shielded by cable clamp made of metal.

> **NOTE**
> For the 15”/19” LCD-mounted type CNC and LCD-mounted type CNC with the is functions, use pin 04 as SG for cabling. Leave pin 08 open.

- Power On sequence

Please turn on the power supply on ETP side first.

- Data of CNC, read / write from ETP

The following CNC data can be read and written on the ETP side:

Signal type	30*i*/31*i*/32*i*- B PMC		
	First PMC	Second PMC	Third PMC
Input signal to the PMC from the machine Note 1	X0 to X127 X200 to X327 X400 to X527 X600 to X727 X1000 to X1127	X0 to X127 X200 to X327 X400 to X527 X600 to X727 X1000 to X1127	X0 to X127 X200 to X327 X400 to X527 X600 to X727 X1000 to X1127
Output signal from the PMC to the machine	Y0 to Y127 Y200 to Y327 Y400 to Y527 Y600 to Y727 Y1000 to Y1127	Y0 to Y127 Y200 to Y327 Y400 to Y527 Y600 to Y727 Y1000 to Y1127	Y0 to Y127 Y200 to Y327 Y400 to Y527 Y600 to Y727 Y1000 to Y1127
Internal relay - User area	R0 to R7999	R0 to R1499	R0 to R1499
Keep relay - User area	K0 to K99	K0 to K19	K0 to K19
Data table	D0 to D9999	D0 to D2999	D0 to D2999
Extra relay	E0 to E9999 Note 2	E0 to E9999 Note 2	E0 to E9999 Note 2
Timer - Variable timer	T0 to T499	T0 to T79	T0 to T79
Counter - Variable counter - Fixed counter	C0 to C399 C5000 to C5199	C0 to C79 C5000 to C5039	C0 to C79 C5000 to C5039

NOTE

1 These addresses are for read only.
2 Shared memory of the multi-path PMC. Programs can read and write the same value from/to this memory.

- Protocol

Only direct command on SNP-X protocol is available in CNC.
ETP also must use the same protocol and the same command only.
The process required for writing data more than 3 bytes is the same as that lass than 2bytes.
Please refer the documents of SNP-X protocol for the detail of SNP-X protocol.

- Combined use of the touch panel on the LCD of the CNC and External Touch Panel

ETP cannot be used together with the touch panel on the LCD (excluding Windows CE compatibles) of the LCD-mounted type CNC.
ETP can be used together with the touch panel on the LCD (excluding Windows CE-enabled compatibles) of the stand-alone type CNC.

> **NOTE**
> To use ETP together with the touch panel on the LCD (excluding Windows CE compatibles) of the stand-alone type CNC, it is necessary to specify the function for the function for touch panel control, and the function for external touch panel interface.

Limitation

(1) Only addresses for the PMC set in parameter No. 11310 can be read and written. It is impossible to perform reading from and writing to more than one PMC at the same time.

(2) ETP is Touch Panel made by DIGITAL Co. Ltd. ETPs which can be connected with CNC are as follows.
- GP-450E
- GP-550T
- GP-550S
- GP-2000 SERIES
- GP-3000 SERIES

Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
3119					TPA			

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#3 TPA When the function for the external touch panel interface is selected, the external touch panel interface connection is:

0: Valid.
1: Invalid.

For an external touch panel (called ETP hereinafter), the RS-232C serial port 2 (JD36A or JD54) on the main board of the CNC is used.
When using ETP, set bit 3 (TPA) of parameter No. 3119 to 0.
By this setting, JD36A or JD54 is used for ETP, regardless of the setting of I/O CHANNEL (I/O device selection) of the existing parameters Nos. 0021 to 0023.
For other I/O devices, use JD56A and so forth.
By the setting above, the settings of the existing parameters Nos. 0100 and 0121 to 0123 become invalid for channel 2 (JD36A or JD54), and the following settings are applied at all times:

- Baud rate : 19200 bps
- Stop bit : 1 bit
- Parity check : Even parity

	#7	#6	#5	#4	#3	#2	#1	#0
13101							TPB	

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 TPB Baud rate used with the external touch panel
0: 19200 bps is always used.
1: The baud rate with the baud rate number set in parameter No. 0123 for channel 2 is used.

As mentioned in the description of bit 3 (TPA) of parameter No. 3119, when TPA is set to 0, the baud rate is always set to 19200 bps.
To allow the baud rate to be changed, set bit 1 (TPB) of parameter No. 13101 to 1.
This allows the baud rate number set in parameter No. 0123 for channel 2 to be used.

> **NOTE**
> Baud rates that can be set may vary depending on the ETP used.

11310	Selection of a PMC that performs read and write operations with an external touch panel

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 3

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

This parameter selects an PMC for read and write operations among three PMCs, which are the first PMC, second PMC, and third PMC.
0, 1 : First PMC
2 : Second PMC
3 : Third PMC

> **NOTE**
> It is impossible to perform read and write operations with more than one PMC at the same time.

13.1.6 Parameter Check Sum Function

Overview

Standard check sum of CNC parameter can be calculated previously and be saved in CNC. In the other hand, a parameter check sum is calculated at CNC power-on. This value is compared with the standard check sum to check whether there is difference between two values.
If there are changed parameters, an alarm will occur at next power-on. So the CNC can be averted from miss operation such as miss setting of parameter and forgetting to correct the parameters changed temporarily.

Explanation

When bit 0 (CKS) of parameter No. 13730 is changed from 0 to 1, the standard check sum is calculated. In the other hand at CNC power-on, a parameter check sum is calculated and its value is compared with standard check sum. If two values are different, alarm DS5340," PARAMETER CHECK SUM ERROR", occurs.

This alarm is canceled by the operation pressing RESET and CAN, but if parameters are not corrected to original value, alarm DS5340 will occur again at next check sum comparison at CNC power-on.
By parameter setting, some parameters can be excluded from check sum. Moreover, there are many parameters that CNC system excludes from check sum. (Refer to excluded parameters in this section)

> **NOTE**
> If bit 7 (CSR) of parameter No. 13730 is 1, alarm DS5340 can be canceled only by RESET key.

Parameter check sum information screen

The parameter check sum information screen shows the value of the standard check sum, the time and date when the standard check sum was calculated, and the value of a check sum calculated at power-on.

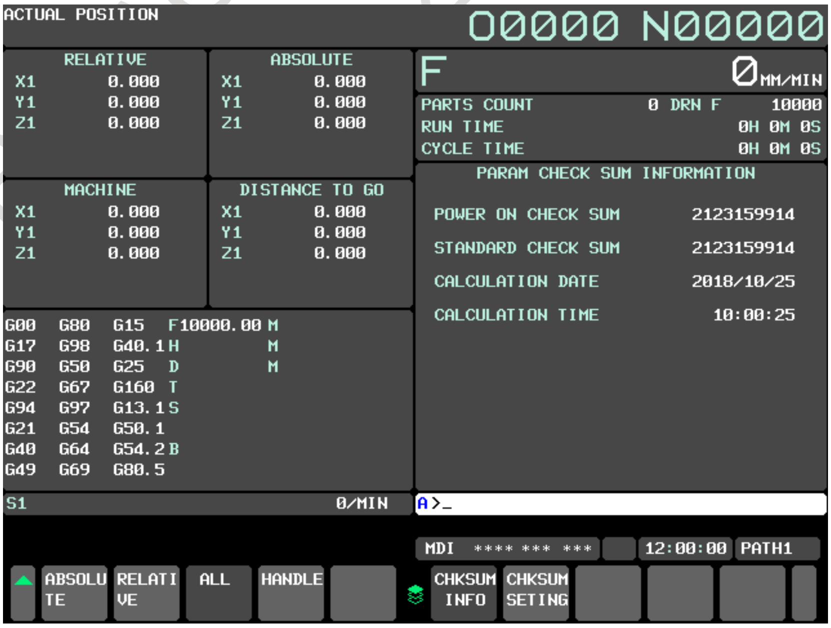


Fig. 13.1.6 (a) Parameter check sum information screen (10.4")

Parameter check sum setting screen

- Setting parameters to be excluded

Setting the number of parameter excluded from check sum on this screen.
A brief explanation about setting data is shown on the bottom of this frame.

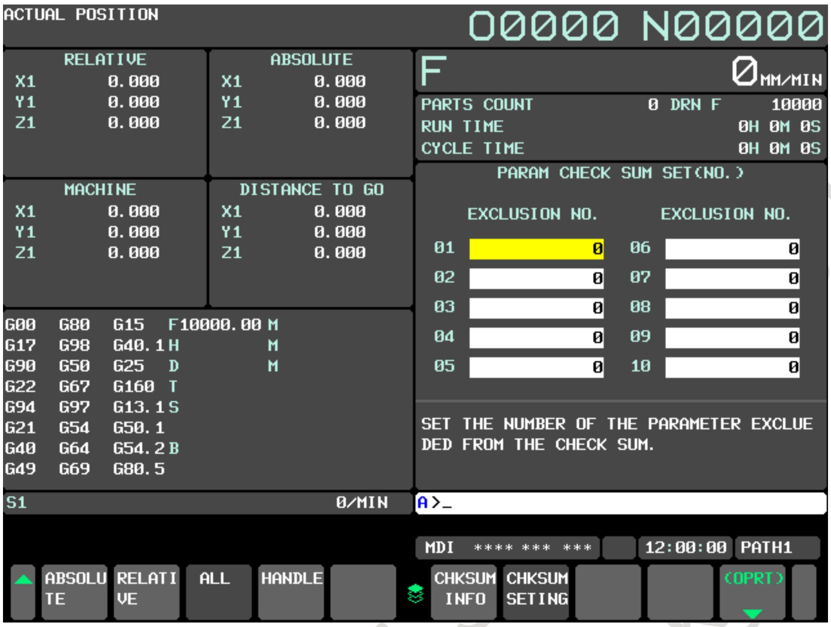


Fig. 13.1.6 (b) Parameter check sum setting (No.) screen (10.4”)

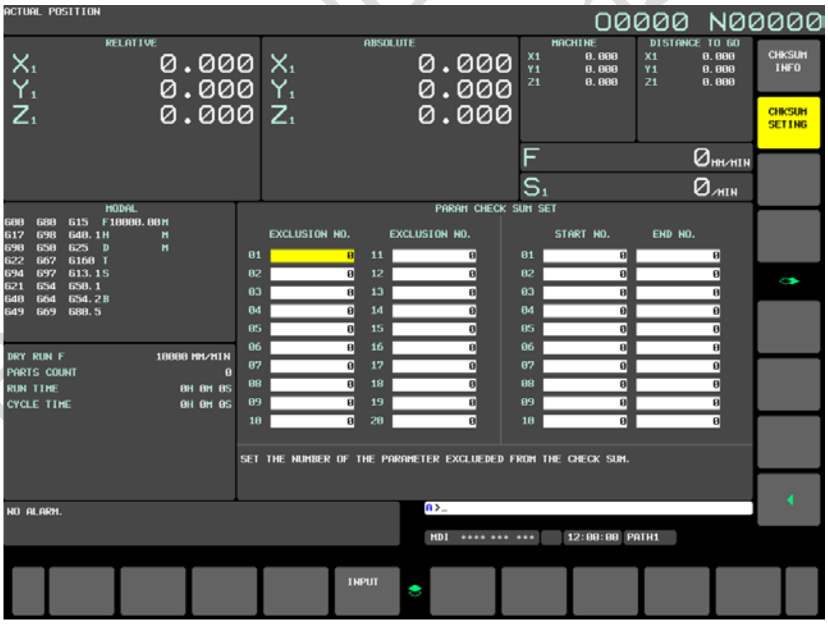


Fig. 13.1.6 (c) Parameter check sum setting screen: number setting (15”)

- Range of excluded parameters

Setting the range of parameters excluded from check sum.
A brief explanation about setting data is shown on the bottom of this frame.

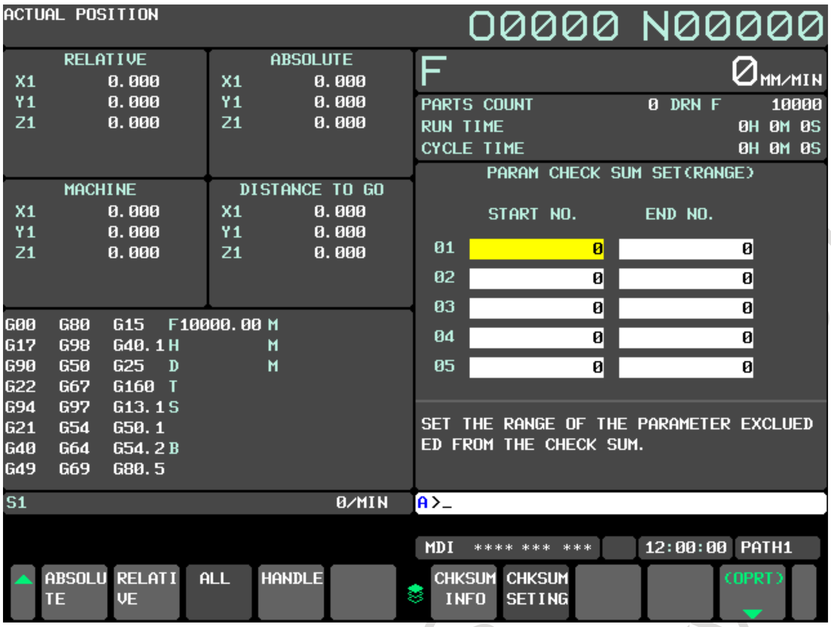


Fig. 13.1.6 (d) Parameter check sum setting (range) screen (10.4”)

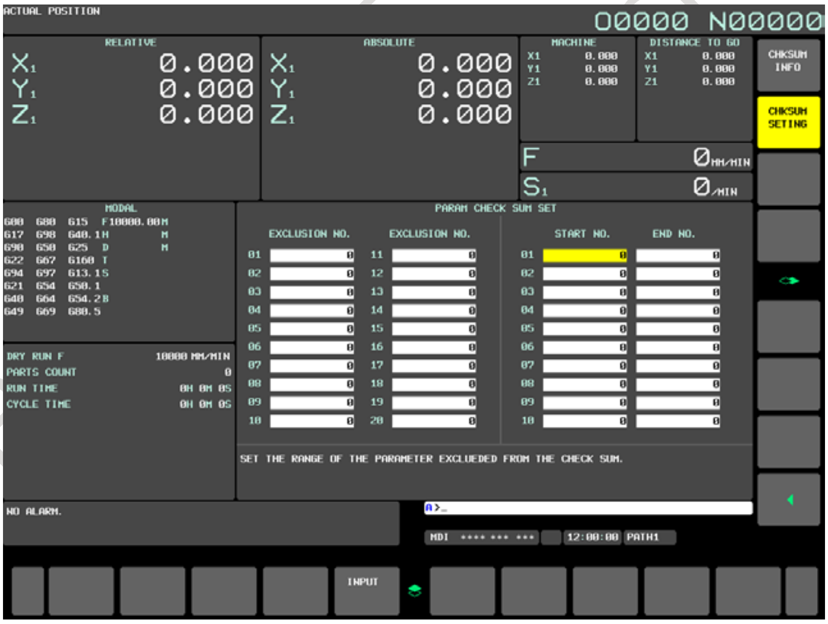


Fig. 13.1.6 (e) Parameter check sum setting screen: range setting (15”)

Operation procedure

The following explains how to display the parameter check sum information screen and how to make settings on the parameter check sum setting screen.

- Displaying parameter check sum information

For the 10.4” display unit, follow the steps below.

1 Press function key SYSTEM.

2 Press continuous menu key several times until soft key [PARAM CHKSUM] appears.

3 Press horizontal soft key [PARAM CHKSUM].

4 Press horizontal soft key [CHKSUM INFO]. The screen in the Fig. 13.1.6 (f) is then displayed:

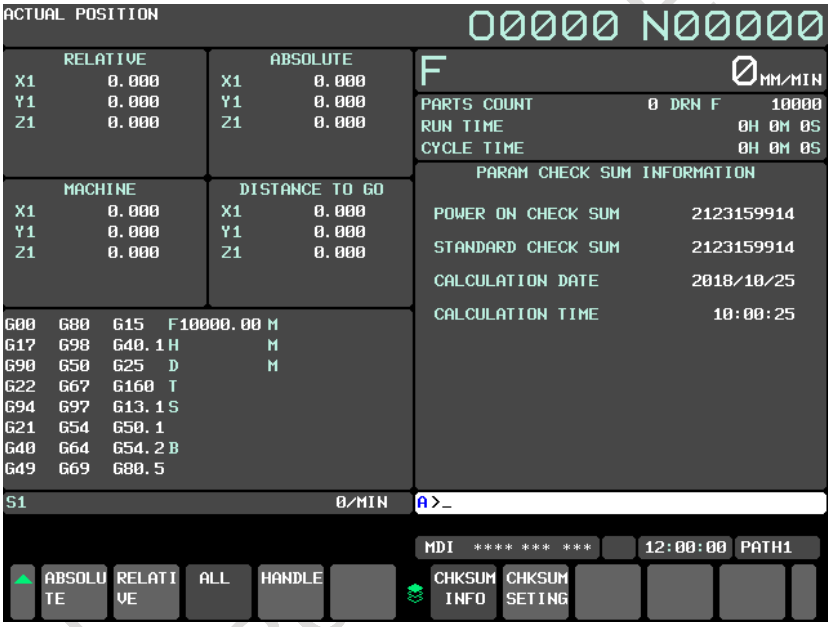


Fig. 13.1.6 (f) Parameter check sum information screen (10.4”)

For the 15” display unit, follow the steps below.

1 Press function key SYSTEM.

2 Press vertical soft key [NEXT PAGE] several times until soft key [PARAM CHKSUM] is displayed.

3 Press vertical soft key [PARAM CHKSUM].

4 Press vertical soft key [CHKSUM INFO]. The screen in the Fig. 13.1.6 (g) is displayed:

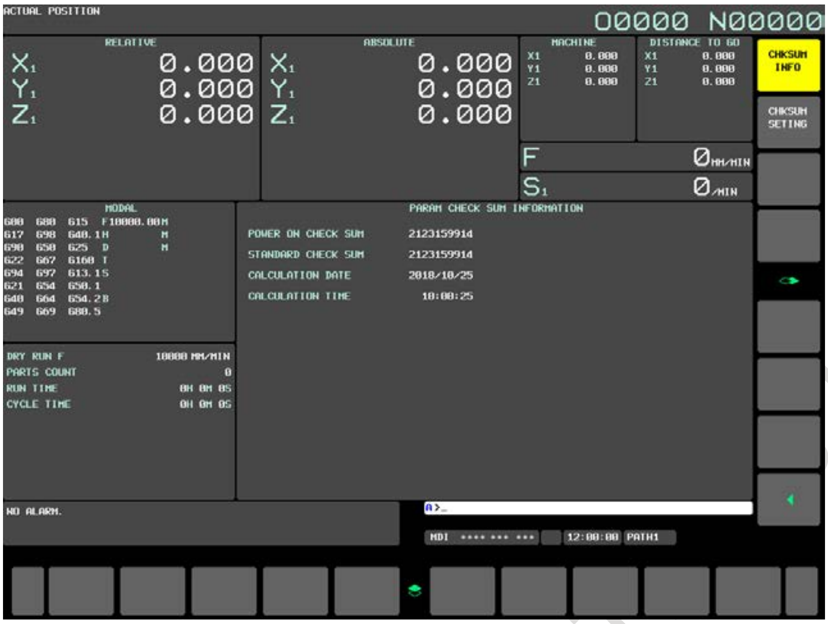


Fig. 13.1.6 (g) Parameter check sum information screen (15")

- Setting parameters to be excluded

For the 10.4" display unit, follow the steps below.

1 Press function key .

2 Press continuous menu key [+] several times until soft key [PARAM CHKSUM] is displayed.

3 Press horizontal soft key [PARAM CHKSUM].

4 Press horizontal soft key [CHKSUM SETING]. The screen in the Fig. 13.1.6 (h) is displayed:

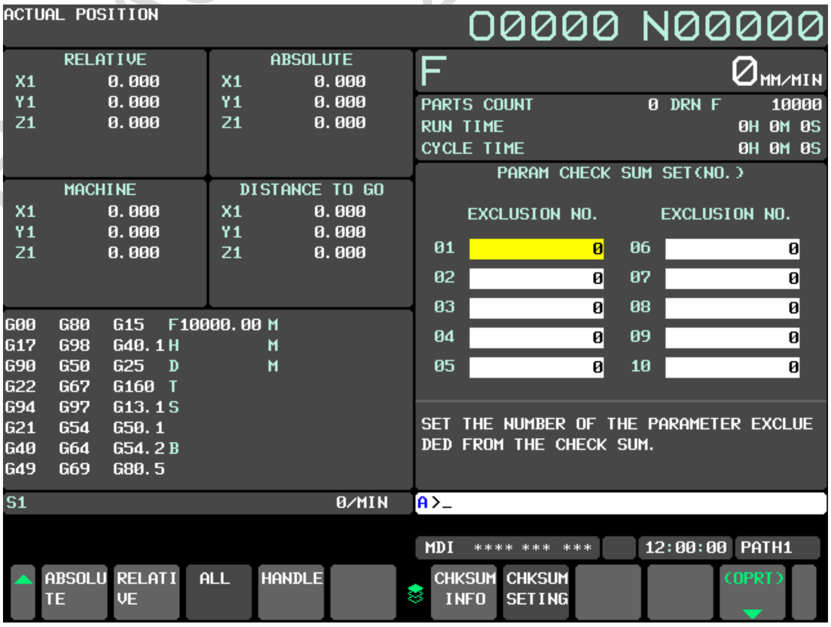
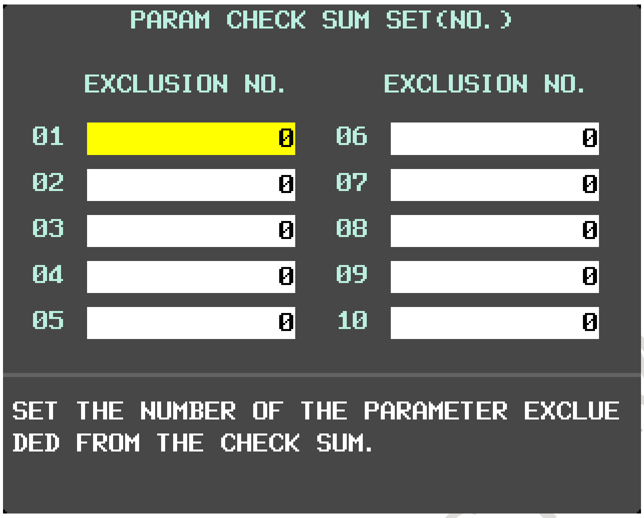


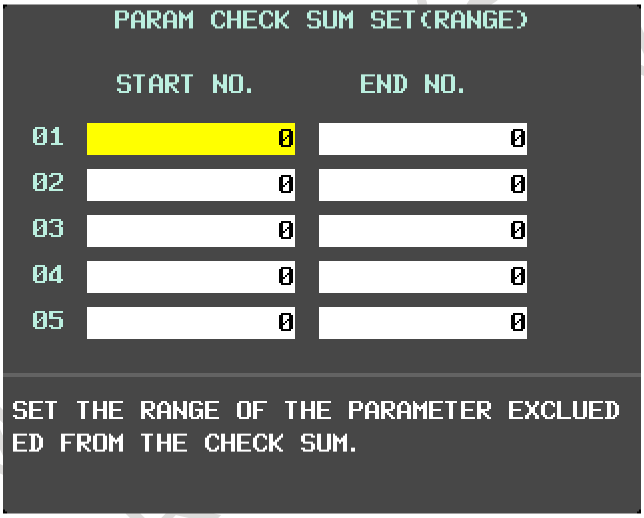
Fig. 13.1.6 (h) Parameter check sum setting screen (10.4")

5 By using page keys and , the parameter check sum setting screen changes as follows:

First page



Third page



6 Move the cursor to a target item.

7 Enter the number of a parameter to be excluded from the check sum, and press the key or soft key [INPUT].

8 When completing the setting of all parameters to be excluded, set bit 0 (CKS) of parameter No. 13730 to 1.

For the 15” display unit, follow the steps below.

1 Press function key 

.

2 Press vertical soft key [NEXT PAGE] several times until soft key [PARAM CHKSUM] is displayed.

3 Press vertical soft key [PARAM CHKSUM].

4 Press vertical soft key [CHKSUM SETING]. The following screen is displayed:

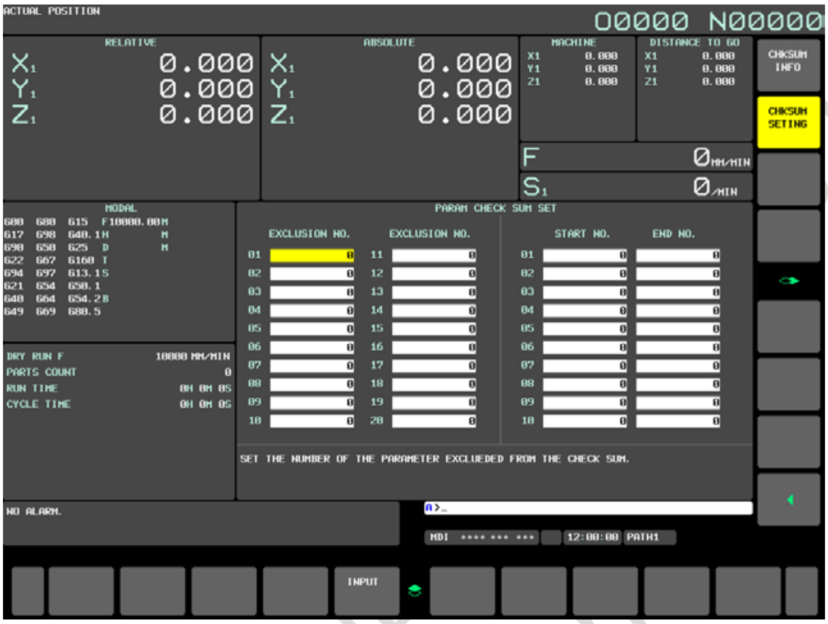
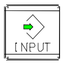


Fig. 13.1.6 (i) Parameter check sum setting screen (15”)

5 Move the cursor to a target item.

6 Enter the number of a parameter to be excluded from the check sum, and press the 

key or soft key [INPUT].

7 When completing the setting of all parameters to be excluded, set bit 0 (CKS) of parameter No. 13730 to 1.

NOTE

1 Items set on this screen correspond to parameter Nos. 13731 to 13770 as shown below. These items can be set also on the parameter screen.
Exclusion Nos. 01 to 20 → Parameters Nos. 13731 to 13750
Exclusion ranges 01 to 10 → Parameters Nos. 13751 to 13770

2 When 0 is set as an exclusion number, it is ignored.

3 Numbers set as the start number and the end number of each exclusion range are also excluded.

4 In a combination of the start No. and end No., if the start No. is greater than the end No. (start No. > end No.), the setting of check sum exclusion numbers becomes invalid.

5 If the start No. equals the end No. (start number = end number), only the set number is excluded.

Excluded parameters

A check sum is not the sum of all parameters but is calculated with particular parameters excluded. The parameters to be excluded satisfy one of the following conditions:

(1) Parameters that can be input by setting input
(2) Parameters of which values may be changed by the system (see Table 13.1.6 (a))
(3) Parameters set on the parameter check sum setting screen

For parameters in (2), see the Table 13.1.6 (a):

Table 13.1.6 (a) Parameters excluded from the check sum

Parameter number	Description
1244	Coordinates of the floating reference position
1320 to 1327	Coordinates at the boundary of stored stroke check 1
1330 to 1348	Chuck and tail stock barrier
3226	Keyword of the key lock for parameters related to dual check safety
4911 to 4914	Spindle speed fluctuation detection
5130	Amount of chamfering in thread cutting cycles G96 and G92
5132 to 5133	Depth of cut and escaping amount of multiple repetitive cycles G71 and G72 (T series)
5134	Clearance value of multiple repetitive cycles G71 and G72
5135	Escaping amount along the X-axis in multiple repetitive cycle G73 (T series)
5136	Escaping amount along the Z-axis in multiple repetitive cycle G73 (T series)
5137	Divide number of multiple repetitive cycle G73 (T series)
5139	Return amount in multiple repetitive cycles G74 and G75 (T series)
5140	Minimum depth of cut in multiple repetitive cycle G76 (T series)
5141	Finishing allowance in multiple repetitive cycle G76 (T series)
5142	Number of repetitions of final finishing in multiple repetitive cycle G76 (T series)
5143	Tool nose angle in multiple repetitive cycle G76 (T series)
5660 to 5668	Flexible synchronization control
5669	Automatic phase synchronization for flexible synchronization control
5670 to 5677	Flexible synchronization control
5680 to 5687	Flexible synchronization control
5690 to 5693	Flexible synchronization control
6581 to 6595	VGA character color number
6750	Integrated value of power-on period
7220 to 7283	Name of general-purpose switch on software operator’s panel
7310	Program restart
8210	Slant angle
8900	PWE
13420	Automatic phase synchronization for flexible synchronization control
13425 to 13436	Automatic phase synchronization for flexible synchronization control
14717	Axis number of the C-axis in simulation (for MGi only)

When the machining condition selecting function is used, the parameters in the Table 13.1.6 (b) to (c) are also excluded from the check sum:

Table 13.1.6 (b) Parameters excluded from the check sum (when the machining condition selecting function is used)

Parameter number	Description
1432	Maximum cutting feedrate for all axes in the look-ahead acceleration/deceleration before interpolation
1769	Time constant for acceleration/deceleration after cutting feed interpolation in the look-ahead acceleration/deceleration before interpolation mode
1772	Acceleration change time of look-ahead bell-shaped acceleration/deceleration before interpolation
1783	Maximum allowable feedrate difference for feedrate determination based on corner feedrate difference
1788	Maximum allowable acceleration change rate in feedrate determination based on acceleration change for each axis
1789	Maximum allowable acceleration change rate in feedrate determination based on acceleration change for each axis (linear interpolation)
1790	Ratio of change time of the rate of change of acceleration in look-ahead smooth bell-shaped acceleration/deceleration before interpolation
13634	Precision level currently selected when AI contour control is used

Table 13.1.6 (c) Parameters for setting the parameter numbers of parameters to be excluded from the check sum (when the machining condition selecting function is used)

Parameter No.
Parameter number set in parameter No. 13628 (parameter number corresponding to item 1 when AI contour control is used)
Parameter number set in parameter No. 13629 (parameter number corresponding to item 2 when AI contour control is used)

NOTE

1 When the machining condition selecting function is not used, these parameters are included in the check sum.

2 The parameters of which numbers are set in parameter Nos. 13628, 13629, 13674, and 13675 are excluded, but parameter Nos. 13628, 13629, 13674, and 13675 are not excluded.

Parameter

11358	Power-On Checksum

[Data type] 2-word
When parameter checksum function is effective, checksum value which is calculated at power-on is set.

0 is set when the parameter checksum function is invalid.

11359	Standard Checksum

[Data type] 2-word
Checksum value which is calculated at when parameter checksum function is changed from invalid to effective is set. It is used as standard value, when checksum is executed at power-on.

0 or last parameter checksum value is set when the parameter checksum function is invalid.

11360	Calculation Data

[Data type] 2-word
The date when the parameter checksum function is changed to effective is set.
The number of 8 digits which show year, month and day is set.
First 4 digits show year. Second 2 digits show month. Last 2 digits show day.

0 or last day when parameter checksum function was changed to effective is set when the parameter checksum function is invalid.

11361	Calculation Time

[Data type] 2-word
The time when the parameter checksum function is changed to effective is set.
The number of 6 digits which show hour, minute and second is set.
First 2 digits show hour. Second 2 digits show minute. Last 2 digits show second.

0 or the last time when parameter checksum function has been changed to effective is set when the parameter checksum function is invalid.

	#7	#6	#5	#4	#3	#2	#1	#0
13730	CSR							CKS

[Data type] Bit

#0 CKS A power-on, a parameter check sum check is:
0: Not performed.
1: Performed.

#7 CSR Alarm DS5340,” PARAMETER CHECK SUM ERROR” is cleared with:
0: 

\+ 

keys.
1: 

key.

13731	Number to be excluded from the NC parameter check sum, 01
13732	Number to be excluded from the NC parameter check sum, 02
13733	Number to be excluded from the NC parameter check sum, 03
13734	Number to be excluded from the NC parameter check sum, 04
13735	Number to be excluded from the NC parameter check sum, 05
13736	Number to be excluded from the NC parameter check sum, 06
13737	Number to be excluded from the NC parameter check sum, 07
13738	Number to be excluded from the NC parameter check sum, 08
13739	Number to be excluded from the NC parameter check sum, 09
13740	Number to be excluded from the NC parameter check sum, 10
13741	Number to be excluded from the NC parameter check sum, 11
13742	Number to be excluded from the NC parameter check sum, 12
13743	Number to be excluded from the NC parameter check sum, 13
13744	Number to be excluded from the NC parameter check sum, 14
13745	Number to be excluded from the NC parameter check sum, 15
13746	Number to be excluded from the NC parameter check sum, 16
13747	Number to be excluded from the NC parameter check sum, 17
13748	Number to be excluded from the NC parameter check sum, 18
13749	Number to be excluded from the NC parameter check sum, 19
13750	Number to be excluded from the NC parameter check sum, 20

[Data type] 2-word path
[Valid data range] 0 to maximum parameter number

These parameters set the numbers of the parameters to be excluded from the check sum in the parameter check sum function.

13751	Start number of the range to be excluded from the NC parameter check sum, 01
13752	Start number of the range to be excluded from the NC parameter check sum, 02
13753	Start number of the range to be excluded from the NC parameter check sum, 03
13754	Start number of the range to be excluded from the NC parameter check sum, 04
13755	Start number of the range to be excluded from the NC parameter check sum, 05
13756	Start number of the range to be excluded from the NC parameter check sum, 06
13757	Start number of the range to be excluded from the NC parameter check sum, 07
13758	Start number of the range to be excluded from the NC parameter check sum, 08
13759	Start number of the range to be excluded from the NC parameter check sum, 09
13760	Start number of the range to be excluded from the NC parameter check sum, 10

13761	Start number of the range to be excluded from the NC parameter check sum, 11
13762	Start number of the range to be excluded from the NC parameter check sum, 12
13763	Start number of the range to be excluded from the NC parameter check sum, 13
13764	Start number of the range to be excluded from the NC parameter check sum, 14
13765	Start number of the range to be excluded from the NC parameter check sum, 15
13766	Start number of the range to be excluded from the NC parameter check sum, 16
13767	Start number of the range to be excluded from the NC parameter check sum, 17
13768	Start number of the range to be excluded from the NC parameter check sum, 18
13769	Start number of the range to be excluded from the NC parameter check sum, 19
13770	Start number of the range to be excluded from the NC parameter check sum, 20

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to maximum parameter number

These parameters specify the range of parameters to be excluded from the check sum in the parameter check sum function. The parameters ranging from the start number to the end number are excluded from the check sum.

NOTE

1. The parameters with the start and end numbers are also excluded.
2. In a combination of start and end numbers, if the start number is greater than the end number (start number > end number), the combination is invalid.
3. If the start and end numbers are the same (start number = end number), the single parameter with that number is excluded.

Alarm and message

Number	Message	Description
DS5340	PARAMETER CHECK SUM ERROR	Because parameters have been modified, the parameter check sum and the standard check sum do not match. Restore the original parameter state, or set a standard check sum again.

13.1.7 Touch Panel Check Signal

Overview

This function outputs a signal to notify the PMC that a virtual MDI key has been pressed. The machine tool builder can use this function for applications in which, for example, a buzzer is sounded after a press of a virtual MDI key is detected.

Explanation

When a virtual MDI key is pressed, the touch panel check signal TPPRS<F0006.0> is output. (Fig. 13.1.7 (a) and Fig. 13.1.7 (b) show the range of virtual MDI keys that cause signal output.)
In related functions, this function operates as follows:

- CNC screen display function
  When the virtual MDI key function is used in the CNC screen display function, the signal is output.
- Macro executor
  When the macro executor is used on the virtual MDI screen, the signal is output.
- C language executor
  While the C language executor screen is being displayed, the signal is not output.

> **NOTE**
> 1 This function is enabled when the virtual MDI function is used.
> 2 When a virtual MDI key is pressed for a short time or when a virtual MDI key is pressed several times successively, the CNC does not sometimes accept the key input even if the signal is output.

Range of virtual MDI keys that cause signal output

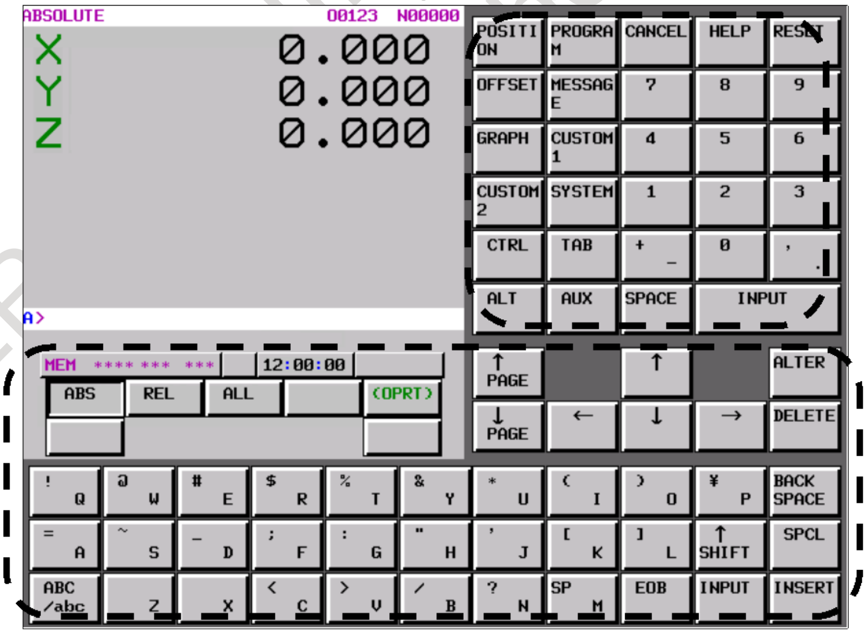


Fig. 13.1.7 (a) When the NC screen is displayed

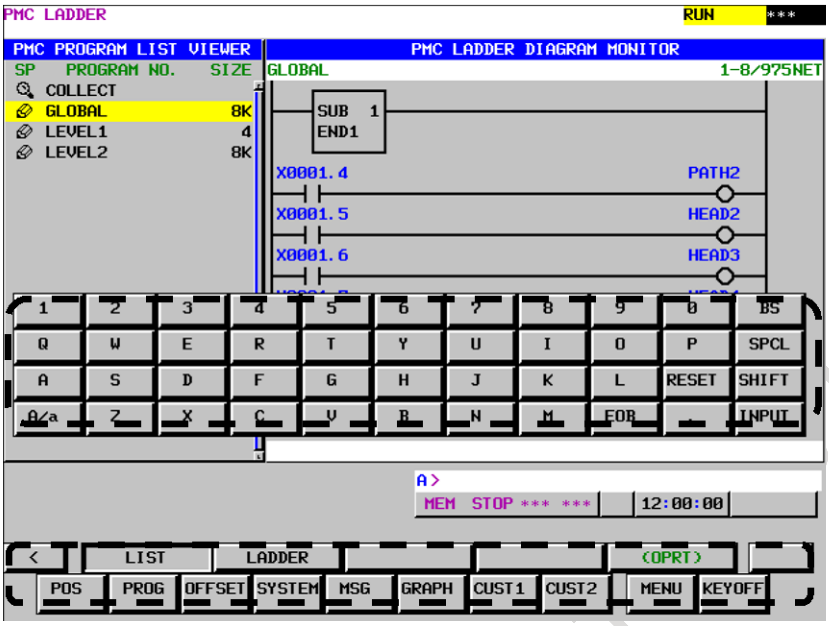


Fig. 13.1.7 (b) When the PMC high-definition screen is displayed

When a virtual MDI key within the range indicated by [dotted box] in Fig. 13.1.7 (a) and Fig. 13.1.7 (b) is pressed, the signal is output.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3207		TPP						

[Input type] Parameter input
[Data type] Bit

#6 TPP When a virtual MDI key is pressed, signal TPPRS <F0006.0> is
0: Not output.
1: Output

Signal

- Touch panel check signal TPPRS<F0006.0>

[Classification] Output signal
[Function] Notifies the PMC that a virtual MDI key has been pressed.
[Operation] The signal is set to 1 when:
- A virtual MDI key is pressed.

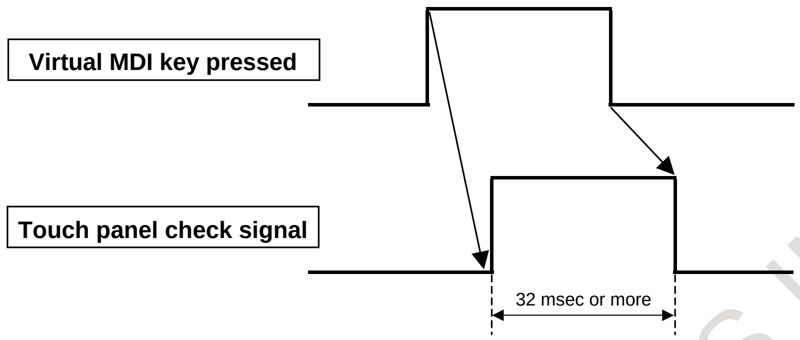
The signal is set to 0 when:
- A virtual MDI key is released.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F0006								TPPRS

- Timing chart

The following shows a timing chart of a press of a virtual MDI key and the touch panel check signal.



- When a virtual MDI key is pressed, the signal is output for 32 msec or longer.
  Read the signal with a ladder program of which cycle period is shorter than 32 msec.
- When a virtual MDI key is held pressed continuously, the signal is kept output.
- There may be a delay from a press of a virtual MDI key until the touch panel check signal is output. A delay may be generated also when the virtual MDI key is released.
- If the time from when a virtual MDI key is pressed until it is released is within 50 msec, the touch panel check signal is not sometimes output.

13.1.8 Changing the Display Language by PMC Signals

Overview

PMC signals can be used to change the display language of the CNC screen. In this function, a dial or switch on the machine operator's panel can also be used to change the language of the CNC screen.

Explanation

To change the display language by the PCM ladder, set display language setting signals LANG1 to LANG7 <G0581.0 to G0581.6> to a number and change display language switch start signal SLANG <G0581.7> from “0” to “1”. This changes the value of parameter No.3281 to the language number that was set in display language setting signals LANG1 to LANG7.
Upon completion of the change, display language switch completion signal FLANG <F0545.0> is set to “1”.

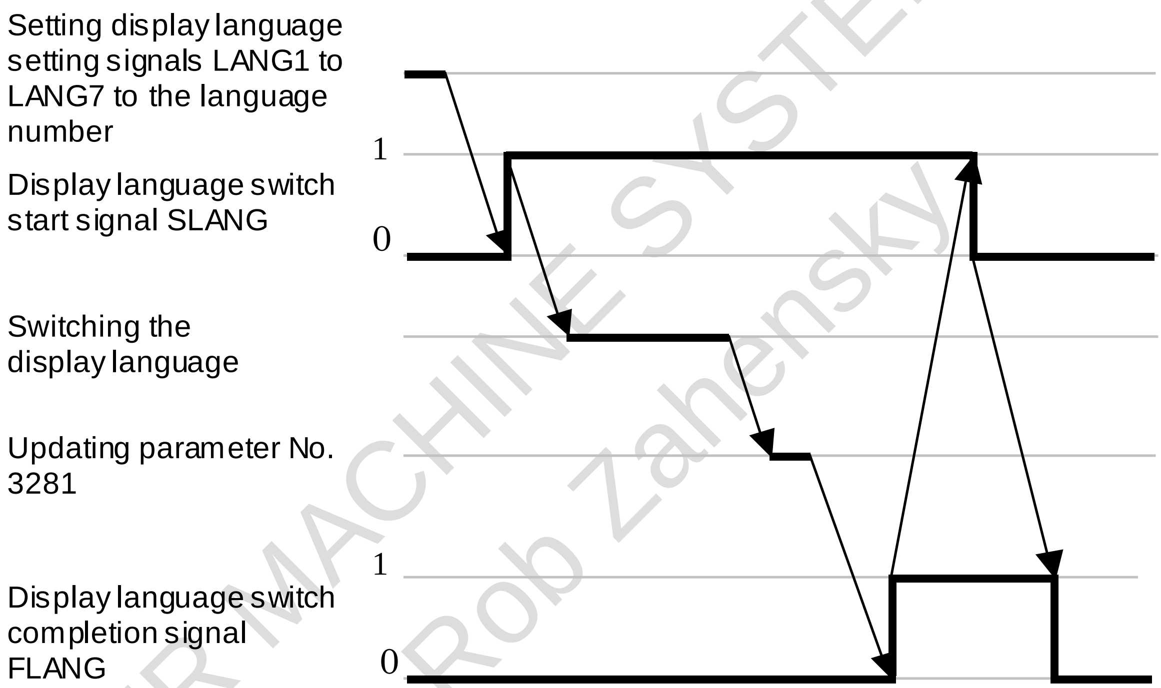
For information on switching the language on the CNC screen, refer to the chapter "Displaying and Switching the Display Language" in OPERATOR’S MANUAL (Common to Lathe System/Machining Center System) (B-64694EN).

NOTE
When the language is selected in the language specification screen, don’t set signal SLANG <G0581.7> to ‘1’.

Time chart

Follow the procedure below to switch the display language.

Set display language setting signals from LANG1 to LANG7 to the language number of the language to be used.
Set display language switch start signal SLANG from “0” to “1”.
The current language is switched to the language with the language number set in display language setting signals from LANG1 to LANG7.
The value of parameter No. 3281 is updated to the value set by display language setting signals from LANG1 to LANG7.
Upon completion of the switching, set display language switch completion signal FLANG is set to “1”.
In the PMC ladder, set display language switch start signal SLANG from “1” to 0 after display language switch completion signal FLANG is set to “1”.
When display language switch start signal SLANG is set to 0, display language switch completion signal FLANG is set to “0”.



Signal

Display language switch start signal SLANG <G0581.7>

[Classification] Input signal

[Function] This signal causes the current language to switch to the language set in display language setting signals LANG1 to LANG7.

[Operation] When this signal is changed from “0” to “1”, the current language switches to the language set in display language setting signals LANG1 to LANG7.

Display language setting signals LANG1 to LANG7 <G0581.0 to G0581.6>

[Classification] Input signal

[Function] These signals set a language number. Available languages and the corresponding input signals are shown in the table.

Language	Language number	LANG7 G0581.6	LANG6 G0581.5	LANG5 G0581.4	LANG4 G0581.3	LANG3 G0581.2	LANG2 G0581.1	LANG1 G0581.0
English	0	0	0	0	0	0	0	0
Japanese	1	0	0	0	0	0	0	1
German	2	0	0	0	0	0	1	0
French	3	0	0	0	0	0	1	1
Chinese (traditional characters)	4	0	0	0	0	1	0	0
Italian	5	0	0	0	0	1	0	1
Korean	6	0	0	0	0	1	1	0
Spanish	7	0	0	0	0	1	1	1
Dutch	8	0	0	0	1	0	0	0
Danish	9	0	0	0	1	0	0	1
Portuguese	10	0	0	0	1	0	1	0
Polish	11	0	0	0	1	0	1	1
Hungarian	12	0	0	0	1	1	0	0
Swedish	13	0	0	0	1	1	0	1
Czech	14	0	0	0	1	1	1	0
Chinese (simplified characters)	15	0	0	0	1	1	1	1
Russian	16	0	0	1	0	0	0	0
Turkish	17	0	0	1	0	0	0	1
Bulgarian	18	0	0	1	0	0	1	0
Rumanian	19	0	0	1	0	0	1	1
Slovak	20	0	0	1	0	1	0	0
Finnish	21	0	0	1	0	1	0	1
Vietnamese	23	0	0	1	0	1	1	1
Indonesian	24	0	0	1	1	0	0	0
Slovenian	25	0	0	1	1	0	0	1

Display language switch completion signal FLANG <F0545.0>

[Classification] Output signal

[Function] This signal indicates that switching to the language set in display language setting signals LANG1 to LANG7 is completed.

[Output cond.] This signal is set to “1” when:

- The display language has been switched.

This signal is set to “0” when:

- Display language switch start signal SLANG is set to “0”.

> **NOTE**
> Even if display language switch start signal SLANG is switched from “0” to “1” when the conditions required for this function are not met, the current language is not switched, but display language switch completion signal FLANG is set to “1”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0581	SLANG	LANG7	LANG6	LANG5	LANG4	LANG3	LANG2	LANG1

	#7	#6	#5	#4	#3	#2	#1	#0
F0545								FLANG

Parameter

3281	Display language

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 21,23 to 25

Select a display language from the following:
- 0 : English
- 1 : Japanese
- 2 : German
- 3 : French
- 4 : Chinese(traditional characters)
- 5 : Italian
- 6 : Korean
- 7 : Spanish
- 8 : Dutch
- 9 : Danish
- 10 : Portuguese
- 11 : Polish
- 12 : Hungarian
- 13 : Swedish
- 14 : Czech
- 15 : Chinese(simplified characters)
- 16 : Russian
- 17 : Turkish
- 18 : Bulgarian
- 19 : Rumanian
- 20 : Slovak
- 21 : Finnish
- 23 : Vietnamese
- 24 : Indonesian
- 25 : Slovenian

If a number not indicated above is set, English is selected.

13.1.9 CNC Screen Dual Display

Overview

When a PC and a CNC are connected via Ethernet or HSSB, the CNC screen can be displayed on the PC and the CNC at the same time by using the CNC screen display function. When the CNC screen display function is started on the PC while a CNC screen is displayed on the display unit of the CNC, the dual display mode is entered.
When the CNC screen display function is ended on the PC, the CNC screen is displayed only on the display unit of the CNC.

In case the PC hangs up for a cause in the dual display mode, it is also possible to forcibly turn off the CNC screen display function on the PC by using Dual display forcible end request signal to end the dual display mode.

Key input operation can be performed on one of the PC and CNC sides, which can be selected.

> **NOTE**
> When the dual display function is being used, the same screen is displayed on the PC and the CNC. The PC performance may have influence on the display speed on the CNC side and slow down the display on the CNC.

Explanation

- Operation

When bit 7 (NS2) of parameter No. 3206 is set to 1, the CNC screen dual display function is enabled.
Then, when the CNC screen display function on the PC side is started, the CNC screen is displayed on both the PC and the CNC .

- Selection of key control

Either the keyboard of the PC or the MDI keys of the CNC become usable for key control.
Key control selection signal CNCKY<G0295.7> is used to select key control.
If CNCKY<G0295.7> is set to "1", the MDI keys of the CNC become usable; if the signal is set to "0", the keyboard of the PC becomes usable.
The reset key, however, is always usable on both the PC and the CNC.
When the CNC is equipped with a touch panel, if bit 5 (S2K) of parameter No. 3206 is set to 1, it becomes possible to select key control by pressing the upper left corner of the screen. In this case, the signal cannot be used to select key control.
When the upper left part of the display screen on the CNC is pressed, the MDI keys become usable; when the upper left part of the CNC screen display on the PC is clicked, the keyboard of the PC becomes usable.

With Key control selection status signal CNCKYO<F0295.7>, key control selection status can be checked to see which key control, key control on the PC or key control on the CNC, is currently selected.

Selection of key control is disabled when the dual display function is not used.

- Data I/O

When bit 0 (PCM) of parameter No. 0300 is set to 1, the data input/output destination is determined by the selection of key control. When the keyboard of the PC is selected, input and output operations on the PC side are enabled; when the MDI keys of the CNC are selected, input and output operations on the CNC side are enabled.

When the USB memory is selected with parameter No. 0020 set to 17, if bit 0 (PCU) of parameter No. 11506 is set to 1, data input/output operation is performed for the PC. If bit 0 (PCU) of parameter No. 11506 is set to 0, data input/output operation is performed for the CNC control unit.

> **NOTE**
>
> If key control selection is changed and so the data input/output destination is changed when the All I/O screen is displayed, the file list displayed on the All I/O screen is not updated automatically. In this case, press soft key [REFRESH] to update the display.

- Forcible end by Forcible end by Dual display forcible end request signal

If the PC hangs up for a cause when the CNC screen dual display function is being performed, the CNC can issue a DI signal to forcibly end the CNC screen display function on the PC.
When C2SEND<G0295.6> is set to "1", the CNC screen display function on the PC is forcibly ended. The status of the forcible end processing is indicated by Dual display forcible end status signal C2SENO<F0295.6>.

Forcible end by Dual display forcible end request signal is invalid when the dual display function is not used.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0300								PCM

[Input type] Parameter input
[Data type] Bit

#0 PCM If a CNC contains a memory card interface, or if the FS0*i* is connected to a PC via HSSB or Ethernet, when the CNC screen display function is started:
0: The memory card interface in the CNC is used.
1: The memory card interface or the hard disk on the PC side is used.
When the CNC screen dual display function is being used, the data input/output destination conforms to the selection of key control.

If a CNC does not contain a memory card interface, the memory card interface on the PC side is used regardless of the setting of this parameter. This parameter is valid only when the CNC screen display function is active.

	#7	#6	#5	#4	#3	#2	#1	#0
3206	NS2		S2K					

[Input type] Parameter input
[Data type] Bit

#5 S2K In CNC screen dual display function,
0: Key control is selected by Key control selection signal <G0295.7>.
1: Key control is selected by pushing at left upper corner on the screen. (Touch panel only)

#7 NS2 CNC screen dual display function is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
11506								PCU

[Input type] Parameter input
[Data type] Bit

#0 PCU If there is a USB memory interface on the CNC side, the USB memory interface used when the CNC screen display function is started via the HSSB interface is on the:
0: CNC side.
1: PC side.

NOTE
1 With the PC function, a USB memory interface on the PC side is used regardless of the setting of this parameter if there is no USB memory interface on the CNC main unit side.
2 This parameter is valid only when the CNC screen display function is active via the HSSB interface.
When setting 1 to this parameter, the CNC screen display function must always be executed via the HSSB interface.
3 When using the CNC screen display function via the Ethernet interface, set 0.
4 When this parameter is set, the power must be turned off before operation is continued.

Signal

Key control selection signal CNCKY <G0295.7>

[Classification] Input signal
[Function] Selects which key control, key control on the PC or on the CNC, is to be enabled.
[Operation] If the signal is set to "0", key control on the PC is enabled.
If the signal is set to "1", key control from the MDI keys on the CNC is enabled.

Dual display forcible end request signal C2SEND <G0295.6>

[Classification] Input signal
[Function] Requests forcible end of the CNC screen display function on the PC side.
[Operation] If the signal is set to "0", no end request is issued.
If the signal is set to "1", an end request is issued.

Key control selection status signal CNCKYO <F0295.7>

[Classification] Output signal
[Function] Indicates which key control, key control on the PC or on the CNC, is currently selected.
[Operation] If this signal is set to "0", key control on the PC side is selected.
If this signal is set to "1", MDI input on the CNC is selected.

Dual display forcible end status signal C2SENO <F0295.6>

[Classification] Output signal
[Function] Indicates the status of end processing performed in response to the end request issued with the dual display forcible end request signal.
[Operation] If this signal is set to "0", the end processing is not yet completed.
If this signal is set to "1", the end processing is completed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0295	CNCKY	C2SEND						

	#7	#6	#5	#4	#3	#2	#1	#0
F0295	CNCKYO	C2SENO						

13.1.10 Twin display function with Ethernet

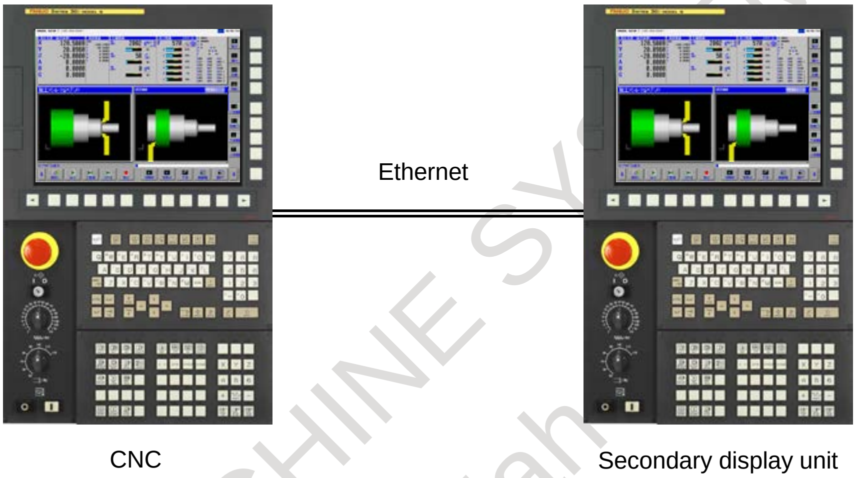
Overview

The twin display function with Ethernet provides the same display and operation as the standard CNC on the secondary display unit for Ethernet connection. (Maximum cable length is 100m.)

The available type of CNC and the kind of Ethernet are as follows.

(1) Type of CNC
  (a) LCD-mounted type CNC
  (b) Stand-alone type CNC

(2) Kind of Ethernet
  (a) Embedded Ethernet with embedded Ethernet port (hereafter referred to as the embedded Ethernet)
  (b) Fast Ethernet (the Ethernet board, the Multi-function Ethernet)



NOTE

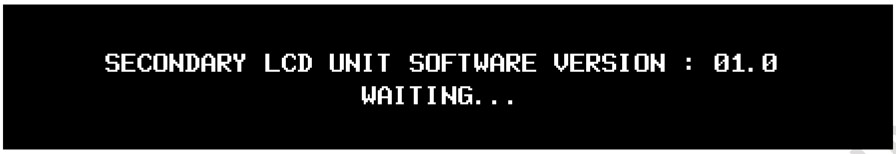
1 The “Twin display function with Ethernet” option is necessary.
2 If the "Twin display function with Ethernet" option is specified, the CNC display function cannot be available at the same time.
3 The display size of the secondary display unit must be the same as the display size of the CNC.
4 The MDI type of the secondary display unit can be different from one of the CNC.
5 CNCs with the personal computer function for Windows XP or Windows CE are not supported.
6 The following screens cannot be displayed on the secondary display unit.
  - BOOT/IPL screen
  - System alarm screen
7 As for the machine operator's panel by the secondary display unit, equip the I/O link and the ladder program for the secondary display unit separately.

Operation

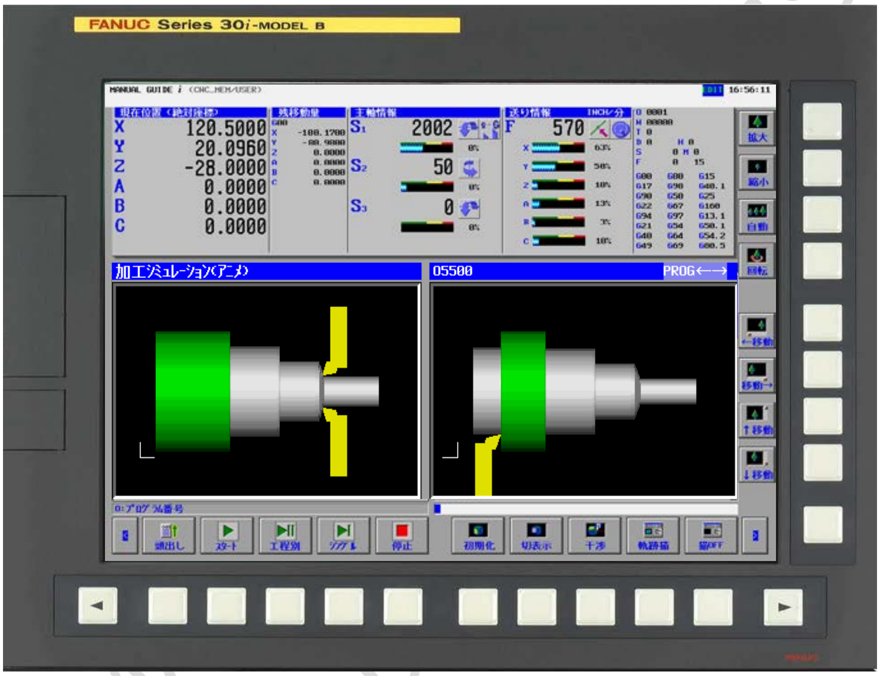
- Start

(1) Turn on the power of the CNC and the secondary display unit.

(2) The following message is displayed on the secondary display unit, then the connection to the CNC is waited.



(3) After the CNC is connected, the same screen of the CNC is displayed on the secondary display unit.



Secondary display unit

> **NOTE**
> The connection settings for the CNC and the secondary display unit are necessary before connecting. Refer to the "Connection settings" later described for details.

- Operation selection for the MDI key and the touch panel

The operation by the MDI key and the touch panel is available on either the CNC or the secondary display unit only.
To select which of them, use the following Key control selection signal CNCKY<G0295.7>.

> **NOTE**
> 1 The reset key is always usable on both the CNC and the secondary display unit.
> 2 It the Ethernet connection is broken during the operation of the secondary display unit, the operation on the secondary display unit and the CNC becomes impossible. Then about 30 seconds later, the operation on the CNC becomes possible.

The related signals are as follows.

Key control selection signal CNCKY <G0295.7>

[Classification] Input signal

[Function] Selects which unit (the CNC or the secondary display unit) is used for the operation by the MDI key and the touch panel.

[Operation] “0” : Secondary display unit
“1” : CNC

Key control selection status signal CNCKYO <F0295.7>

[Classification] Output signal

[Function] Shows which unit (the CNC or the secondary display unit) is selected for the operation by the MDI key and the touch panel at present.

[Operation] “0” : Secondary display unit
“1” : CNC

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0295	CNCKY							

	#7	#6	#5	#4	#3	#2	#1	#0
F0295	CNCKYO							

- Usage of memory card or USB memory in the secondary display unit

When you use a memory card or a USB memory in the secondary display unit, set the following NC parameters. (When you use the CNC side, set the former value to the NC parameters.)

0020	I/O CHANNEL: Input/output device selection, or interface number for a foreground input device

[Input type] Setting input
[Data type] Byte
[Valid data range] 0 to 17

Table for settings and input/output devices	
Setting	**Description**
4	Memory card or USB memory in the secondary display unit.

	#7	#6	#5	#4	#3	#2	#1	#0
0300								PCM

[Input type] Setting input
[Data type] Bit

#0 PCM In case of the secondary display unit for Ethernet connection,
0: A memory card or a USB memory in the CNC is effective.
1: A memory card or a USB memory in the secondary display unit is effective.

> **NOTE**
> The limitations in case of using a memory card or a USB memory in the secondary display unit are as follows:
> 1 The supported functions are the data input/output (read/punch) only. The following functions are not available.
>   - DNC operation, schedule operation and external subprogram call (M198)
>   - Memory card program operation/editing.
>   - Servo Guide
>   - FANUC LADDER-III
>
>   Moreover, the C-language executor and the macro executor cannot access a memory card or a USB memory in the secondary display unit.
> 2 In case of displaying the program folder screen, select the [MEMORY CARD] in [DEVICE CHANGE] even if a USB memory is used.
> 3 If both a memory card and a USB memory are inserted in the secondary display unit, the first inserted device can be used only.

- Notes for connection

When there is a problem about the Ethernet connection, one of the following situations may happen.

(1) The display stops at the waiting screen (the screen of 2) of ”Start” earlier described) for connection to the CNC.
Check the following items.
(a) Start of the CNC
(b) Connection of network
(c) Response from the CNC (Refer to the “Response check of CNC to be connected” of ”Settings & Maintenance” for the secondary display unit” later described)
(d) Ethernet settings for the CNC and the secondary display unit (Refer to the “Connection settings” later described)

(2) The following “CONNECTION FAILED” message is displayed.

The error information is displayed in the following format inside parentheses.
(ERROR: Error No.: Detail1: Detail2)
Handle this error as follows. Detail 1 and 2 show error numbers of the socket error or the protocol error that occurred while communicating. If the problem is not fixed, inform the detail 1 and 2 to FANUC.

Error No.	Meaning	Error handling
-16	Socket error	Check your network connection (Ethernet cable or Ethernet board etc.). If the problem is not fixed, inform the detail 1 and 2 to FANUC.
-17	Protocol error	Inform the detail 1 and 2 to FANUC.
-18	Other error	Inform the detail 1 and 2 to FANUC.

Settings and maintenance

- Connection settings

The following settings are necessary to connect the CNC with onto the secondary display unit.

Settings of the CNC side

Set the IP address and the subnet mask for the CNC, and the TCP port number for the FOCAS2/Ethernet function.
Set also the following NC parameter.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3206	NS2							

[Input type] Parameter input
[Data type] Bit

#7 NS2 Twin display function with Ethernet is:
0: Disabled.
1: Enabled.

Settings of the secondary display unit side

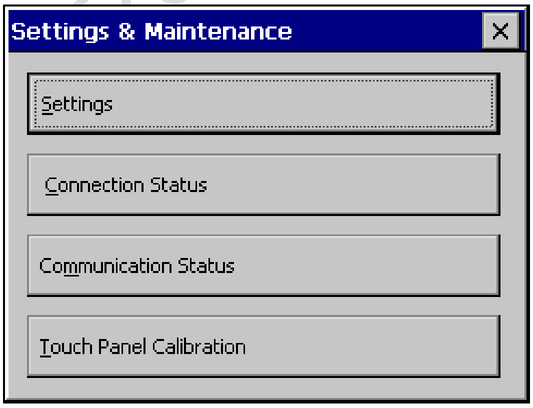
Set the IP address and the subnet mask for the secondary display unit. Also set the IP address and the TCP port number for the CNC.

> **NOTE**
> 1 For the CNC, set the same IP address and TCP port number as "Settings of the CNC side".
> 2 As for the setting method, refer to the “Settings” of ““Settings & Maintenance” for the secondary display unit” later described.

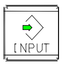
“Settings & Maintenance” for the secondary display unit.

Display the "Settings & maintenance" menu for the settings and maintenance of the secondary display unit by either of the following operations.

- Keep pressing ALT +[S] key more than 2 seconds.
- Keep pressing the upper right corner (within 3cm) on the touch panel display more than 3 seconds.



“Settings & Maintenance” menu

To select the menu button, press on the touch panel or press 

key after select by the cursor key on the MDI keyboard.
As for the operations of each menu items, refer to the following table.

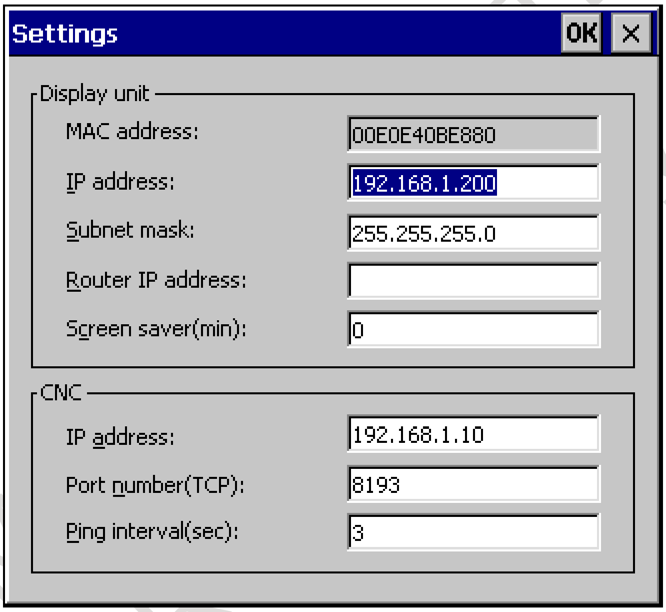
Button name	Reference
Settings	Settings
Connection Status	Response check of CNC to be connected
Communication Status	Communication status with CNC
Touch Panel Calibration	Calibration of the touch panel

To close this menu, press the  button on the screen or press 

key on the MDI keyboard.

- Settings

Select the "Settings" in the "Settings & maintenance" menu, then the setting screen for the Ethernet and the backlight of the secondary display unit will be displayed.



As for the setting items, refer to the following tables (1) and (2).

(1) Display unit: Settings for the secondary display unit

Item	Description	Initial value
MAC address	MAC address (cannot be changed)	-
IP address	IP address	192.168.1.200
Subnet mask	Subnet mask	255.255.255.0
Router IP address	Router IP address (Set if necessary)	None
Screen saver(min)	After this time passes without input of the keys and the touch panel, the backlight is turned off. (Minute) (Range: 0 to 127) In case of "0", the backlight is not turned off.	0

(2) CNC: Settings for the CNC to be connected

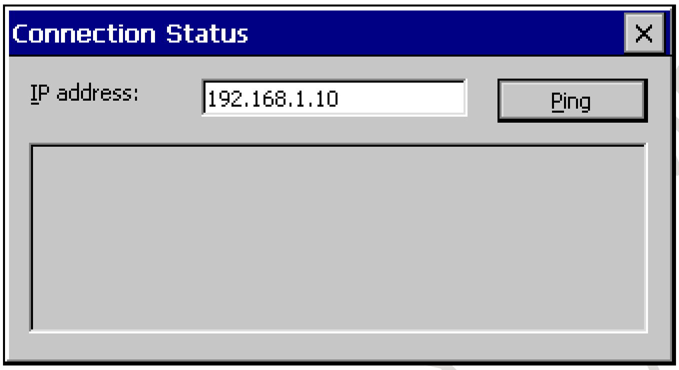
Item	Description	Initial value
IP address	IP address	192.168.1.10
Port number (TCP)	TCP port number (Range: 5001 to 65535)	8193
Ping interval (sec)	Interval time of periodical response check for the CNC (second) (Range : 0 to 120)	3

By pressing the OK button on the screen or INPUT key on the MDI keyboard, the setting values are saved and the "Settings & maintenance" menu comes back.

By pressing the × button on the screen or AUX key on the MDI keyboard, the setting values are ignored and the "Settings & maintenance" menu comes back.

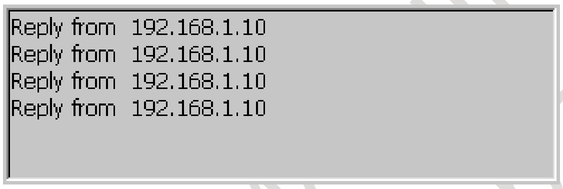
- Response check of CNC to be connected

Select the "Connection Status" in the "Settings & maintenance" menu, then the screen for the response check of CNC to be connected will be displayed.

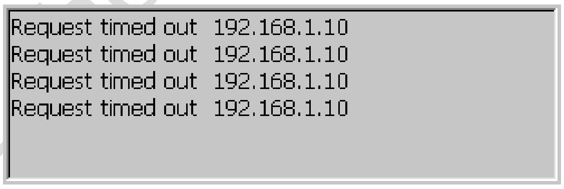


Input an IP address (Ex.: 192.168.1.10) of CNC to check its response, then press the Ping button or INPUT key on the MDI keyboard.

The checking by the PING command is run 4 times, and the result is displayed.



The response is received.

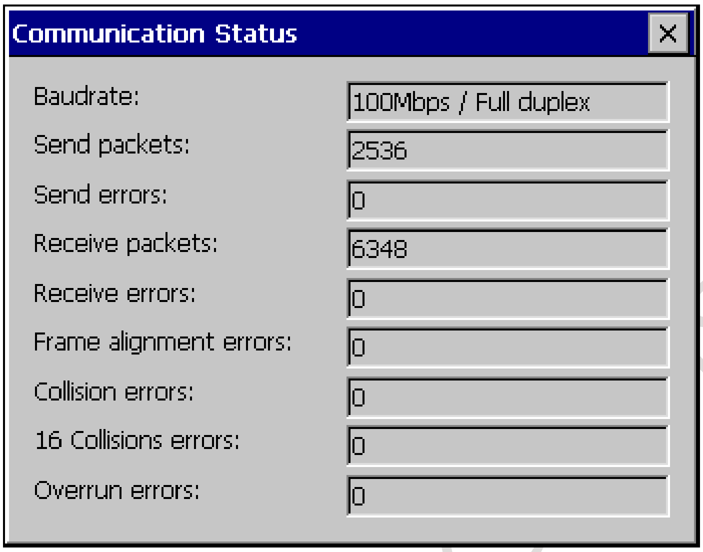


The response is not received.

By pressing the × button on the screen or AUX key on the MDI keyboard, the "Settings & maintenance" menu comes back.

- Communication status with CNC

Select the "Communication Status" in the "Settings & maintenance" menu, then the screen for the communication status with CNC will be displayed.



The communication with CNC is continued while displaying the screen and the data of communication status is updated.

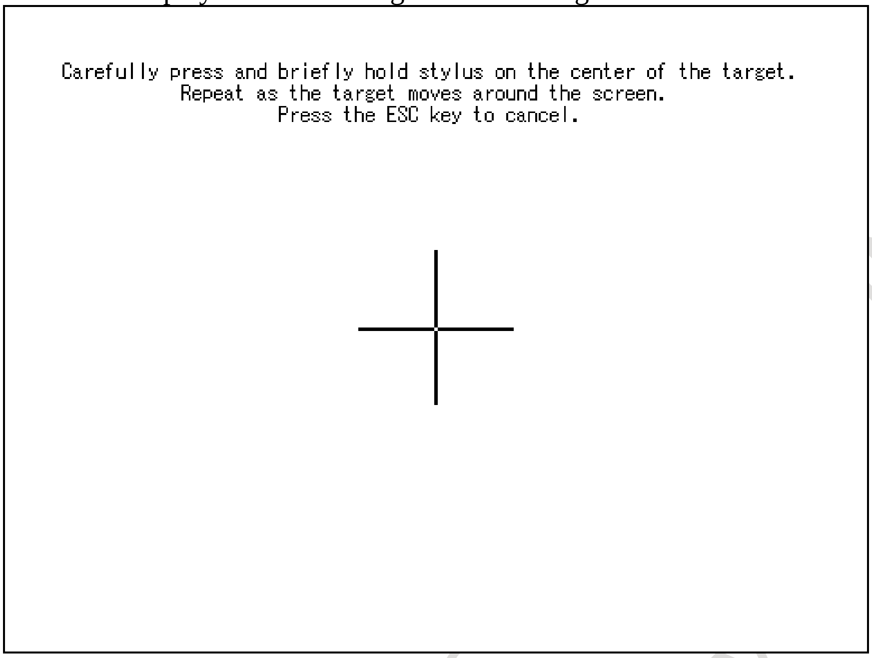
The displayed items are as follows:

Item	Description
Baudrate	Communication rate and mode. When the communication is not established, "------" is displayed.
Send packets	Number of send packets
Send errors	Number of send errors
Receive packets	Number of receive packets
Receive errors	Number of receive errors
Frame alignment errors	Number of frame alignment errors
Collision errors	Number of collision errors
16 Collisions errors	Number of 16 consecutive collision errors
Overrun errors	Number of overrun errors

By pressing the X button on the screen or AUX key on the MDI keyboard, the "Settings & maintenance" menu comes back.

- Calibration of the touch panel

Select the "Touch Panel Calibration" in the "Settings & maintenance" menu, then the calibration screen for the touch panel will be displayed. Follow the guidance messages shown in the screen.



After the calibration is finished, the "Settings & maintenance" menu comes back.

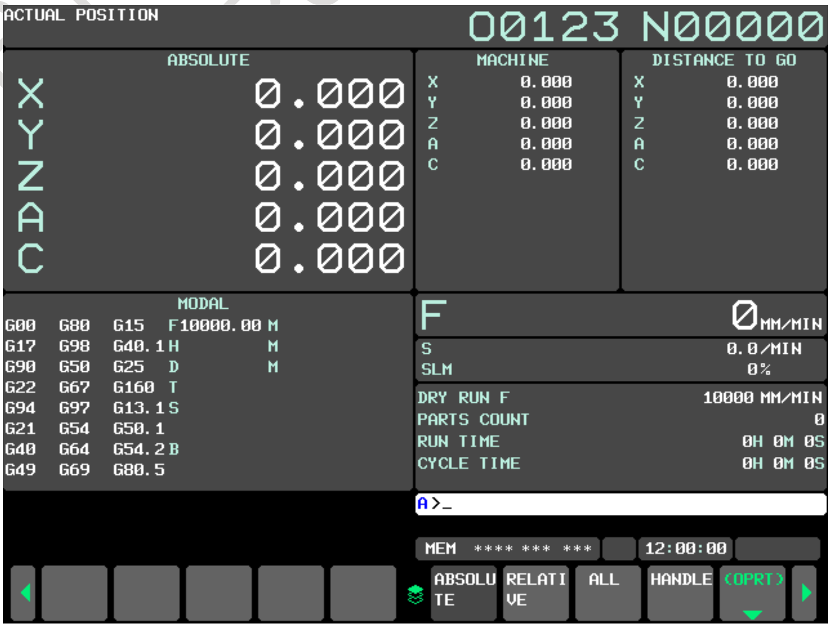
13.1.11 Speed Display Function of a Milling Tool with Servo Motor

Overview

Any servo motor axis can be selected to display its speed considering gear ratio.

Explanation

The screen display can be switched between the speed of the milling axis and the spindle speed by using an input signal SDPC <Gn038.5>.



←(*)

Fig. 13.1.11 (a)

(*)→

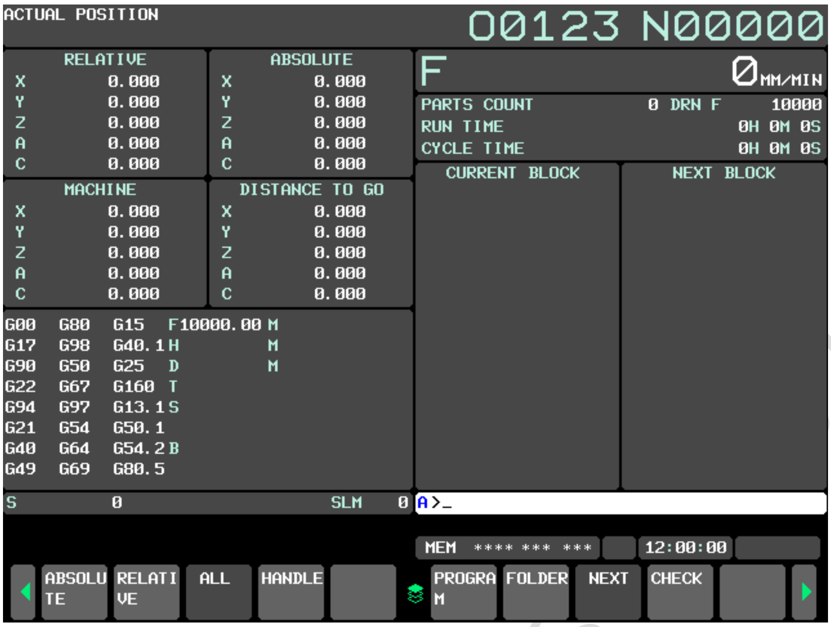
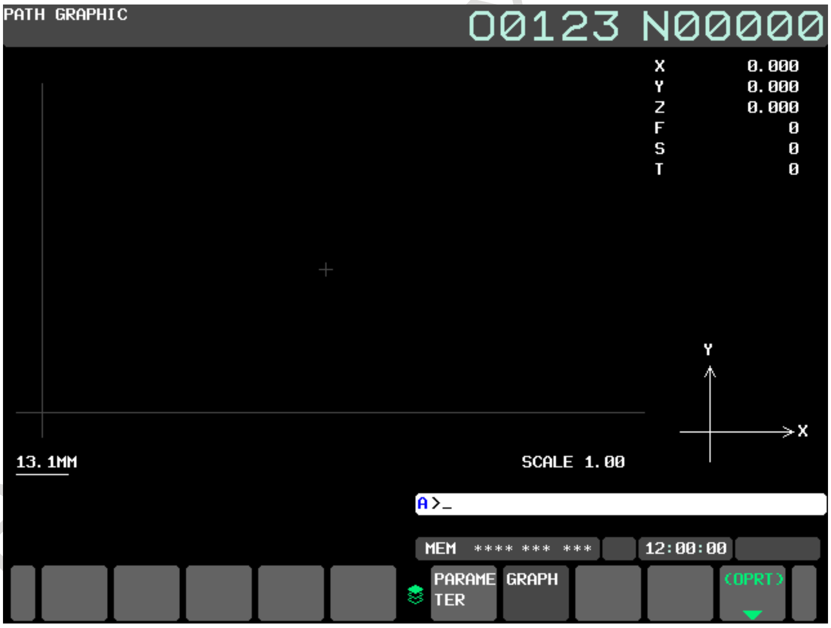


Fig. 13.1.11 (b)



←(*)

Fig. 13.1.11 (c)

- Switching to the display of the speed of a milling tool with a servo motor

With the speed display change signal SDPC<Gn038.5>, the speed of a milling axis or the spindle speed can be selected and indicated in the part marked by (*) in the Fig. 13.1.11 (a) to Fig. 13.1.11 (c).

- Displaying the speed of a milling tool with a servo motor

As the speed of a milling axis, the value obtained by multiplying the servo motor speed by the gear ratio is indicated.

$$\text{Gear ratio} = \frac{\text{Number of gear teeth on the servo motor axis side (parameter No.1898)}}{\text{Number of gear teeth on the milling axis side (parameter No.1899)}}$$

To select the target servo motor axis, set a controlled axis number as the number of a servo motor axis used as the axis of a milling tool (parameter No. 1895).

> **NOTE**
> 1 If parameter Nos. 1898 and 1899 are not set, the gear ratio is assumed to be 1:1.
> 2 If an axis that uses a linear motor is selected, the speed cannot be displayed.
> 3 Even when the speed of a milling axis is displayed, the actual speed of feed per revolution follows the speed of the spindle.

Signal

Speed display change signal SDPC<Gn038.5>

[Classification] Input signal

[Function] Selects the speed to be displayed: the speed of the milling axis or the spindle speed.

[Operation] If this signal is set to "1", the speed of the milling axis is displayed.
If this signal is set to "0", the spindle speed is displayed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn038			SDPC					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3105						DPS		

[Input type] Parameter input
[Data type] Bit path

#2 DPS The actual spindle speed is:
0: Not displayed
1: Displayed

1895	Servo motor axis number used for a milling tool

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes

This parameter sets the servo motor axis number used for displaying the speed of a milling tool that incorporates a servo motor.

1898	Number of gear teeth on the servo motor axis side

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 1 to 9999

This parameter sets the number of servo motor axis gear teeth used for displaying the speed of a milling tool that incorporates a servo motor.

> **NOTE**
> This parameter is valid when a non-zero value is set in parameter No. 1895.

1899	Number of gear teeth on the milling axis side

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 1 to 9999

This parameter sets the number of milling axis gear teeth used for displaying the speed of a milling tool that incorporates a servo motor.

> **NOTE**
> This parameter is valid when a non-zero value is set in parameter No. 1895.

13.1.12 Screen Switching by Mode

Overview

One of the screens displayed using function key 

such as the program edit screen and the program check screen can be selected according to the mode. When a mode switches to other mode, the screen selected last in that mode is displayed.

There are five modes, and immediately after the power is turned on in each mode, a screen is selected as follows:

MEM	:	Program check screen (for MEM)
EDIT	:	Program edit screen (for EDIT)
MDI	:	Program edit screen (for MDI)
TJOG/THND	:	Program edit screen
Other modes	:	Program edit screen

Mode switching and screen examples

- Mode switching after power-on

When the MEM mode is selected and function key 

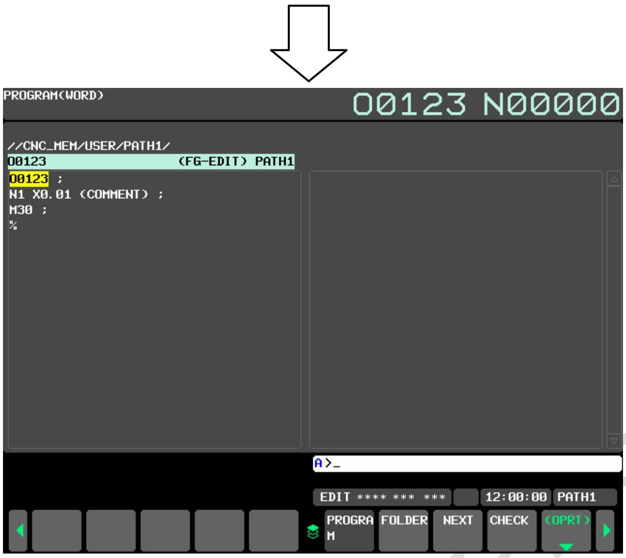
is pressed at power-on, the program check screen is displayed.
When the mode is changed to the EDIT mode, the program edit screen is displayed.

- Mode switching after screen operation

When soft key [FOLDER] is pressed in the EDIT mode, the program list screen is displayed.
When the mode is changed to the MEM mode, the program check screen is displayed.
Then, when the mode is changed to the EDIT mode, the program list screen is displayed.

MEM mode

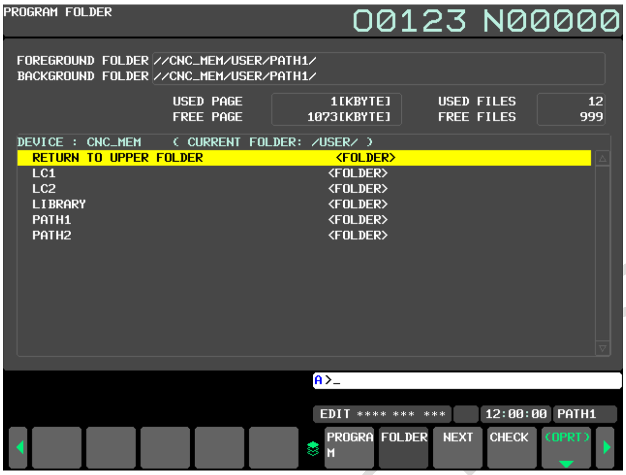
EDIT mode


PROGRAM(WORD)
O0123 N00000
//CNC_MEM/USER/PATH1/
O0123 (FG-EDIT) PATH1
O0123 ;
N1 X0.01 (COMMENT) ;
M30 ;
%
A>_
EDIT **** *** *** 12:00:00 PATH1
PROGRAM
FOLDER
NEXT
CHECK
(OPRT)

- Mode switching after screen operation

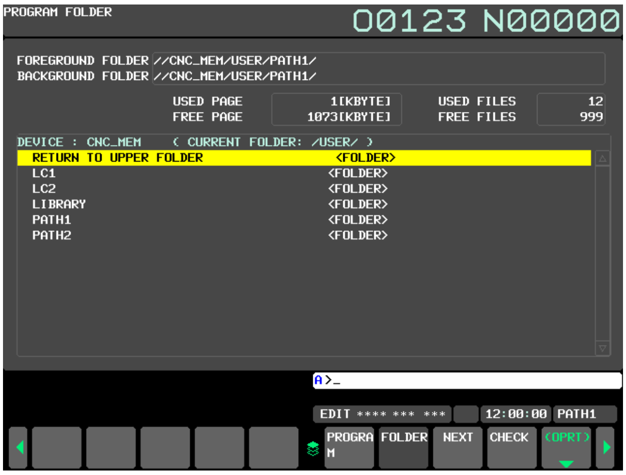
When soft key [FOLDER] is pressed in the EDIT mode, the program list screen is displayed.
When the mode is changed to the MEM mode, the program check screen is displayed.
Then, when the mode is changed to the EDIT mode, the program list screen is displayed.

EDIT mode



MEM mode

EDIT mode



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11302	CPG							

[Input type] Parameter input
[Data type] Bit

#7 CPG PROG function screen selection is:
0: Not changed according to the CNC mode.
1: Changed according to the CNC mode.

13.1.13 Screen Switching at Path Switching

Overview

When switching from one path to another has been made, the same screen and soft keys as displayed for the previous path can be displayed.
Since the same screen is displayed even after path switching, the user no longer has to change the screen display.
It is also possible to change the screen display to the screen that was selected most recently for each path.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3208			PSC					

[Input type] Parameter input
[Data type] Bit

#5 PSC When the path is switched based on the path switch signal:
0: The screen display is switched to the last selected screen of the path.
1: The same screen as for the path before switching is displayed.
If this parameter is set to 0, the screen selected for each path is stored.
Therefore, when path switching to a path is made, the screen selected last for that path is displayed.
Even if the path is changed, FANUC PICTURE and manual guide ***i*** screen does not change to other screen because these screens are common to each path.

Therefore, different behavior will exist together in case of PSC=0, like as some screen stay in the screen and other screen changed to another screen when the path is changed.
Use with PSC=1 is recommended, so that the screen should not change by the path change on all screens.

13.1.14 Screen Erasure Function and Automatic Screen Erasure Function

Overview

Keeping the same characters displayed in the same positions on the screen for a long time will shorten the life of the LCD.
To prevent this, the CNC screen can be erased. The screen erasure function allows the user to perform a key operation to erase the screen. The automatic screen erasure function erases the screen automatically when there has been no key operation for a parameter-set period of time.

Screen erasure function

When 0 is set in parameter No. 3123, the CNC screen can be erased by pressing the CAN key and any function key (such as POS or PROG) at the same time. The CNC screen can be displayed again by pressing any function key.

Automatic screen erasure function

When there has been no key operation for a time (in minutes) set in parameter No. 3123, the CNC screen is erased automatically. The CNC screen is displayed again by pressing a key.

- Screen erasure by the automatic screen erasure function

If the following conditions are all satisfied for the time (in minutes) set in parameter No. 3123, the CNC screen is erased.
Conditions for automatically erasing the CNC screen

- Parameter No. 3123 $\neq$ 0
- None of the following key operations is performed.
  MDI keys
  Soft keys
  External key input
- No alarm is issued.

- Redisplay of the screen by the automatic screen erasure function

If one of the following conditions is satisfied when the CNC screen is off, the CNC screen is displayed again:
Conditions for redisplaying the CNC screen

- One of the following key operations is performed.
  MDI keys
  Soft keys
  External key input
- An alarm is issued.

- Screen erasure by using the CAN key + function key

When a non-zero value is set in parameter No. 3123, the screen is not erased with the CAN key and a function key.

- Set time

Only the time set in parameter No. 3123 for path 1 is valid.

- Alarm in another path

When an alarm is issued in any of the paths, the screen is not erased.

Signal

Screen erasure disable signal *CRTOF<G0062.1>

[Classification] Input signal

[Function] This signal enables or disables the screen erasure function/automatic screen erasure function.

[Operation] 0: Enables the screen erasure function/automatic screen erasure function.

1: Disables the screen erasure function/automatic screen erasure function.

Automatic screen erasure status in-progress signal ERTVA<Fn006.2>

[Classification] Output signal

[Function] This signal indicates that the CNC screen is being erased by the automatic screen erasure function.

[Operation] This signal is set to “1” when:

- The CNC screen is erased by the automatic screen erasure function.

This signal is set to “0” when:

- One of the following key operations is performed:
  MDI key
  Soft key
  External key input
- An alarm is issued.
- The operation mode is changed.
- The automatic screen erasure disable signal *CRTOF <G0062.1> is set to “1”.

NOTE
This signal is not output when the screen is erased by pressing the key and any function key.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0062							*CRTOF	

	#7	#6	#5	#4	#3	#2	#1	#0
Fn006						ERTVA		

Parameter

3123	Time required before a screen saver is activated

[Input type] Setting input

[Data type] Byte path

[Unit of data] min

[Valid data range] 0 to 127

After a time (in minutes) set in parameter No. 3123 passes without key operation, the NC screen is erased automatically. Pressing a key causes the NC screen to reappear.

NOTE
1 Setting 0 disables automatic screen erasure.
2 This function cannot be used together with manual screen erasure. If 1 or a larger value is set in this parameter, manual screen erasure is disabled.

13.1.15 Screen Hard Copy Function

Overview

This function converts screen information displayed on the CNC into bit map format data and output it to a memory card or USB memory. Once output, bit map format data can be displayed and edited on a personal computer.

Explanation

- Start/cancellation methods

The screen hard copy function is started by pressing and holding down key SHIFT for five seconds or by changing the hard copy execution request signal HCREQ <G0067.7> from "0" to "1". The function can be canceled by pressing key CAN or by changing the hard copy cancellation request signal HCABT <G0067.6> to "1".
While the screen hard copy function is in progress, the hard copy execution status signal HCEXE <F0061.3> is "1", and upon completion, it is set to 0. When a screen hard copy cancellation request is received, the hard copy cancellation request reception signal HCAB2 <F0061.2> is set to "1" and remains in the 1 state until the hard copy function is started again or a reset is made.

- Acquisition and output of screen data

When the screen hard copy function is started, the function acquires screen data and, immediately after the acquisition, outputs bitmap format data to, for example, a memory card inserted into the LCD unit. While screen data is being acquired, the screen remains stationary for a few seconds. Acquired screen data can be output from the memory card screen. Also, while screen data is being output, "OUTPUT" blinks in the status display.

- Screen data file names

Bit map format screen data files created by this function are assigned the names below:

In case of bit 4 (HCN) of parameter No. 3301 is 0, file name starts with "HDCPY000.BMP" and increments the number to 999 as following.
"HDCPY000.BMP" (Name of a file to be first output to a memory card etc. after power-on)
"HDCPY001.BMP" (Name of a file to be second output to a memory card etc. after power-on)
:
:
"HDCPY999.BMP" (Name of a file to be 1000 output to a memory card etc. after power-on)
If the screen hard copy function is executed after a file with "HDCPY999.BMP" is output, the file name returns to "HDCPY000.BMP". However, if a file with the same file name as the one to be output when the screen hard copy function is executed exists on the memory card, alarm SR1973 is issued. If the capacity of the memory card is exceeded, alarm SR1962 is issued.
In either case, screen data is not output, so that either the existing file needs to be deleted or the memory card needs to be replaced with a new one.
In case of using a USB memory, if the name of a file to be output during execution of screen hard copy already exists in the memory, the file is overwritten. If the capacity of the USB memory is exceeded, alarm SR1932 is issued. In this case, please delete unnecessary files or use a new USB memory.

In case of parameter HCN is 1, file name id defined by the date and the execute number of screen hard copy function on that day as following.
"YYMMDD00.BMP" (Name of a file to be first output to a memory card etc. on that day)
"YYMMDD01.BMP" (Name of a file to be second output to a memory card etc. on that day)
:
:
"YYMMDD99.BMP" (Name of a file to be 100 output to a memory card etc. on that day)

(YY(00~99) means the year, MM(01~12) means the month and DD(01~31) means the day)

(Example)

If screen hard copy is executed on 2013/04/17 in the first time, bit map format file is created with the name of “13041700.BMP”.

If the screen hard copy function is executed after a file with “YYMMDD99.BMP” is output, the file name returns to "YYMMDD00.BMP". However, if a file with the same file name as the one to be output when the screen hard copy function is executed exists on the memory card, system searches number which does not exist and output the file with that number. If all number files (00~99) exist on the memory card, alarm SR1973 is issued. If the capacity of the memory card is exceeded, alarm SR1962 is issued.

In either case, screen data is not output, so that either the existing file needs to be deleted or the memory card needs to be replaced with a new one.

In case of using a USB memory, if the name of a file to be output during execution of screen hard copy already exists in the memory, system searches the file number which does not exist on the memory and output the file with that number, same as memory card. If all number files (00~99) exist in the memory, the file is overwritten. If the capacity of the USB memory is exceeded, alarm SR1932 is issued. In this case, please delete unnecessary files or use a new USB memory.

Limitation

- Screens whose hard copies cannot be made

Hard copies of the BOOT screen, the IPL screen, and the system alarm screen cannot be made.

- Foreground I/O devices

During DNC operation, for example, screen data cannot be output while a foreground I/O device is in use.

- Canceling the hard copy function

If the hard copy function is canceled before a hard copy is completed, an incomplete bit map file of data that has been output is created.

- CNC screen display function

If CNC screen display function is working, the screen hard copy function is disabled. In this case, bit map format data can be made by screen copy function of CNC screen display function. Please refer to the operator’s manual of CNC screen display function.

- Output folder

The screen data is output to the root folder of the memory card or the USB memory.

Signal

Hard copy cancellation request signal HCABT <G0067.6>

[Classification] Input signal

[Function] This signal requests the cancellation of the screen hard copy function.

[Operation] When the signal is set to “1”, the control unit operates as follows:

- Cancels the screen hard copy function if it is being executed.
- Does nothing if the screen hard copy function is not being executed. Note that changing HCREQ <G0067.7> from “0” to “1” when HCABT <G0067.6> is “1” does not cause the screen hard copy function to be executed.

> **NOTE**
> If changing this signal from “0” to “1”, set it to “0” on the PMC after the signal is maintained in the 1 state for at least 64 msec.

Hard copy execution request signal HCREQ <G0067.7>

[Classification] Input signal
[Function] This signal requests the execution of the screen hard copy function.
[Operation] When the signal is changed from “0” to “1”, the control unit operates as follows:

- Starts creating a hard copy.

> **NOTE**
> If changing this signal from “0” to “1”, set it to “0” on the PMC after the signal is maintained in the “1” state for at least 64 msec. If making another hard copy execution request, wait until the signal is maintained in the “0” state for at least 64 msec.

Hard copy cancellation request reception signal HCAB2 <F0061.2>

[Classification] Output signal
[Function] This signal notifies that a screen hard copy cancellation request is made.
[Operation] This signal is set to “1” in the following case:

- HCABT <G0067.6> is set to “1” or key CAN is pressed, so that the CNC receives this as a screen hard copy cancellation request.

This signal is set to “0” in the following cases:

- A reset is made.
- The screen hard copy is started again.

Hard copy execution status signal HCEXE <F0061.3>

[Classification] Output signal
[Function] This signal notifies that the screen hard copy function is being executed.
[Operation] This signal is set to “1” in the following case:

- The screen hard copy function is started by changing HCREQ <G0067.7> from “0” to “1” or by pressing and holding down key SHIFT for five seconds.

This signal is set to “0” in the following cases:

- The screen hard copy function is completed.
- The screen hard copy function is canceled.

Timing charts for input/output signals are shown below Fig. 13.1.15 (a) shows a timing chart of the screen hard copy function when it terminates normally, and Fig. 13.1.15 (b) shows a timing chart of the screen hard copy function when it is canceled and restarted.

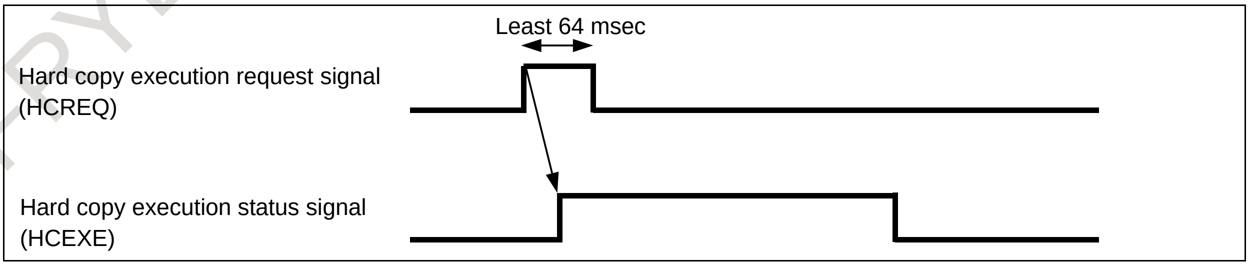


Fig. 13.1.15 (a) When the screen hard copy function terminates normally

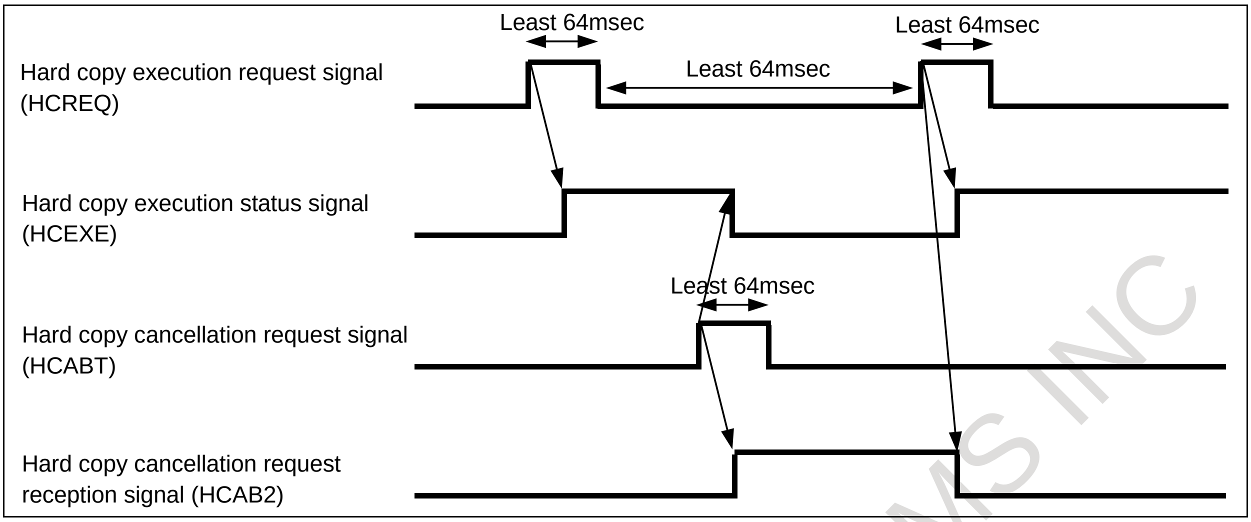


Fig. 13.1.15 (b) When the screen hard copy function is canceled and restarted

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0067	HCREQ	HCABT						

	#7	#6	#5	#4	#3	#2	#1	#0
F0061					HCEXE	HCAB2		

Parameter

0020	I/O CHANNEL : Input/output device selection, or interface number for a foreground input device

[Input type] Setting input
[Data type] Byte
[Valid data range] 0 to 35

Set the interface number for a foreground I/O device.

If enabling the screen hard copy function (by setting bit 7 (HDC) of parameter No. 3301 to 1), set this parameter to 4 (memory card) or 17 (USB memory).

	#7	#6	#5	#4	#3	#2	#1	#0
3301	HDC			HCN				H16

[Input type] Parameter input
[Data type] Bit path

#0 H16 Bit map data of screen hard copies uses:
0: 256 colors.
1: 16 colors.

#4 HCN Bit map file name of the screen hard copy is:
0: "HDCPYxxx.BMP"
(xxx(000~999) means the number of screen hard copy after power-on).
1: "YYMMDDxx.BMP"
(YY(00~99) means the year, MM(01~12) means the month, DD(01~31) means the day and xx(00~99) means the number of screen hard copy in that day)

#7 HDC A screen hard copy function is:
0: Disabled.
1: Enabled.

Only the value of this parameter that is set for path 1 is effective.

13.1.16 Actual Speed Display Axis Selection Signals

Overview

These signals specify whether the speed of each axis is added to the display of actual speed.
The signals can be used when, for example, the movement speed of the slave axis is not added to the display of actual speed during synchronous operation.
The signals are valid for reading of the actual speed of the control axis in the PMC window function (function code 26).

> **NOTE**
> The actual speed obtained by reading of the actual speed of the control axis in the PMC window function (function code 26) is a value used for screen display and cannot be used for control.

Signal

Actual speed display axis selection signals *ACTF1 to *ACTF8 <Gn580>

[Classification] Input signal

[Function] These signals specify whether the movement speed of each axis is added to the display of actual speed.

[Operation] The movement speed of the axis for which 1 is set is not added to the display of actual speed.

*ACTFx :x=1 . . . The speed of the first axis is not added.
x=2 . . . The speed of the second axis is not added.
x=3 . . . The speed of the third axis is not added.
: :

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn580	*ACTF8	*ACTF7	*ACTF6	*ACTF5	*ACTF4	*ACTF3	*ACTF2	*ACTF1

> **NOTE**
> These signals can be used together with bit 3 (NDF) of parameter No. 3115, which specifies whether the movement speed of a selected axis is considered during calculation of actual speed.

13.1.17 Custom Macro Variable Name Expansion 31 Characters

Overview

This function expands the names of custom macro common variables #500 to #549 to a maximum of 31 characters in length.

> **NOTE**
> This function is an optional function.

Explanation

A name of up to eight characters can be assigned to 50 custom macro common variables #500 to #549 using the following command.

SETVN n[VAR500,VAR501,VAR502,……] ;

See Chapter II-16 in the OPERATOR'S MANUAL (B- 64694EN).

If this function is used, a custom macro common variable can have a variable name of up to 31 characters. Up to 31 characters can be displayed on the macro variable screen when:

- The full-screen display of the custom macro variable screen is performed on the 10.4-inch display.
- The custom macro variable screen is displayed on the 15-/19-inch display.

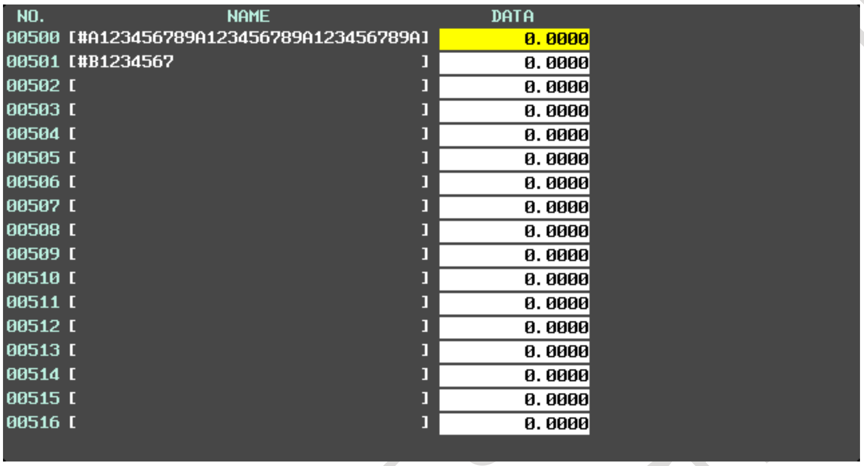


Fig. 13.1.17 (a) 31-character display

In the following cases, custom macro common variable names of up to 11 characters are displayed. If a macro variable name is longer than 11 characters, only the 11 characters are displayed and the rest is represented by "~".

- When the small-screen display of the custom macro variable screen is performed on the 10.4-inch display.

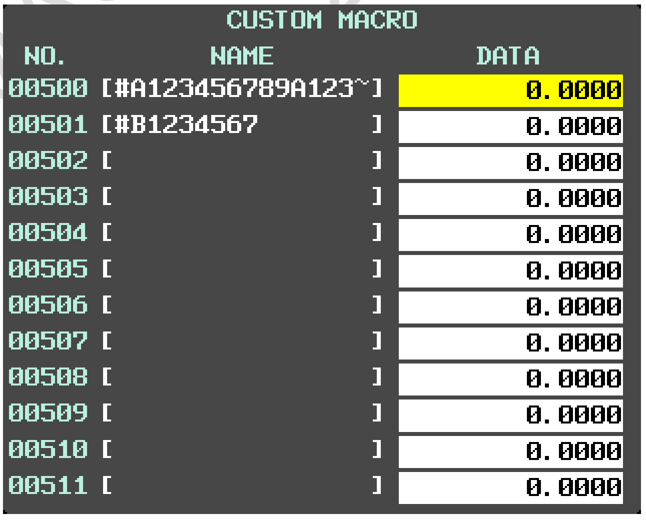


Fig. 13.1.17 (b) 11-character display

Display procedure

1. Press the function key .
2. Press the continuous menu key to display soft key [MACRO].
3. Press the soft key [MACRO].

10.4-inch display unit

The custom macro variable screen below is displayed.
In small screen display, custom macro common variable names of up to 11 characters are displayed.

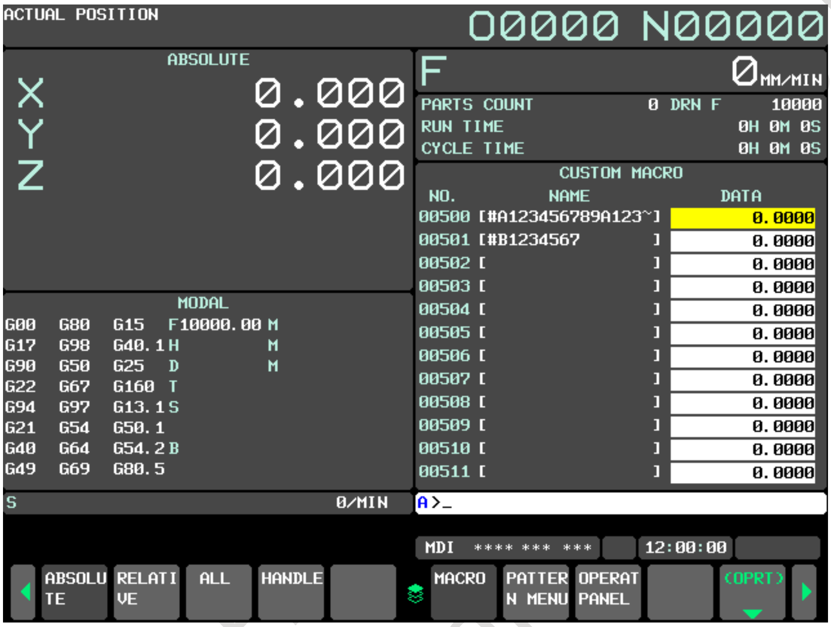


Fig. 13.1.17 (c) 10.4-inch display (small screen display)

Press soft key [MACRO] again to change the custom macro variable screen to full screen display. In full screen display, custom macro common variable names of up to 31 characters are displayed.

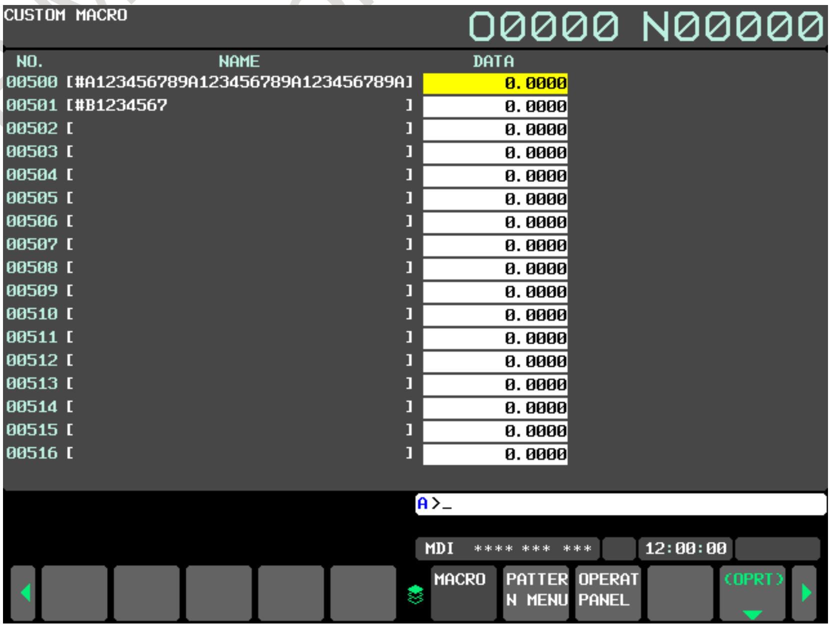


Fig. 13.1.17 (d) 10.4-inch display (full screen display)

NOTE

1. Bit 4 (SCM) of parameter No. 11355 specifies whether small screen display or large screen display is selected in the initial state of the custom macro screen during power-on.
2. If the pattern data input function is enabled and the comment of pattern data input is set to be displayed on the right side, variable names of up to 10 characters are displayed in small screen display.
3. If the macro compiler & executor function is enabled and custom macro common variables #500 to #549 and P-CODE macro common variables are set to be sharable, variable names of up to 31 or 11 characters are displayed on the execution macro variable screen, conversational macro variable screen, and auxiliary macro variable screen, as on the custom macro screen.

15-inch display unit

The custom macro variable screen below is displayed. Custom macro common variable names of up to 31 characters are displayed.

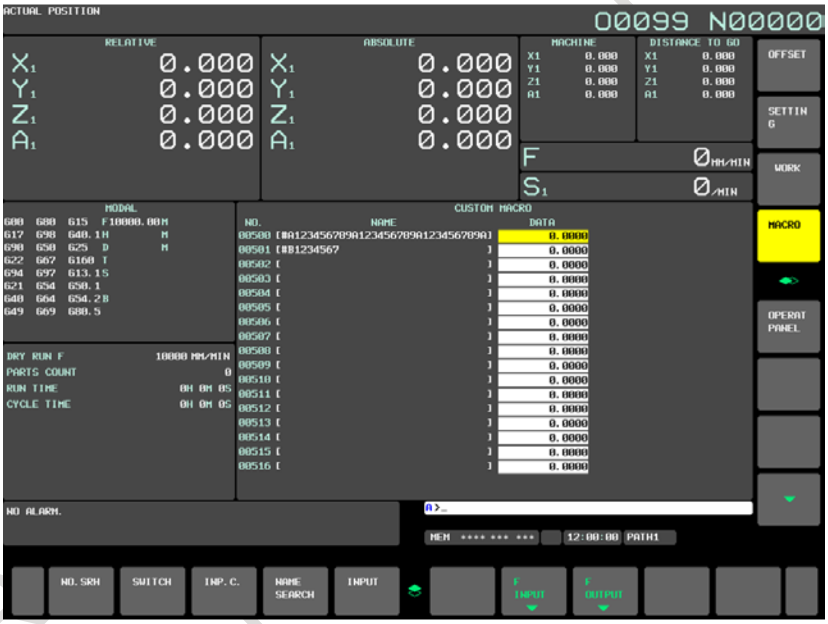


Fig. 13.1.17 (e) 15-inch display

NOTE

If the macro compiler & executor function is enabled and custom macro common variables #500 to #549 and P-CODE macro common variables are set to be sharable, variable names of up to 31 or 11 characters are displayed on the execution macro variable screen, conversational macro variable screen, and auxiliary macro variable screen, as on the custom macro screen.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3207			VRN					

[Input type] Parameter input
[Data type] Bit

#5 VRN On the custom macro variable screen, the variable names of common variables #500 to #549 are:
0: Not displayed.
1: Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
11355				SCM				

[Input type] Parameter input
[Data type] Bit

#4 SCM In the initial state, the custom macro screen is:
0: A small screen display.
1: A full screen display.

13.1.18 Switching the Axis Name of an Axis Type Alarm

Overview

If an axis type alarm occurs, the type, number, and message of the alarm as well as the name of the axis on which the alarm occurred are displayed on the alarm screen. The axis name is usually the axis name set in parameter No. 1020, but bit 3 (DAA) of parameter No. 11368 can be set to switch the axis name to the one set in parameter No. 3132.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11368					DAA			

[Input type] Parameter input
[Data type] Bit

#3 DAA The axis name used with axis type alarms is one set using parameter No.:
0: 1020.
1: 3132.

> **NOTE**
> 1 Even when this parameter is 1, an axis name set in parameter No. 1020 is used if the value of parameter No. 3132 is 0.
> 2 If an extended axis name is in use, only the first letter in it is replaced.

Limitation

Even when bit 3 (DAA) of parameter No. 11368 is 1, the axis name to be displayed on the operation history screen and the alarm history screen is the one set in parameter No. 1020.

13.1.19 Periodic Maintenance Screen

Overview

The periodic maintenance screen function provides guidelines as to when consumables such as the backup battery of the control unit and the backlight of the LCD unit are to be replaced and checked if the lifetimes of these items have been set in advance.

Signal

Periodic maintenance lifetime warning signal LIFOVR<Fn093.0>

[Classification] Output signal

[Function] Reports that a lifetime warning is issued on the periodic maintenance screen.

[Operation] This signal is set to "1" when any of the items set on the periodic maintenance screen enters the lifetime warning state.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn093								LIFOVR

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8900					PLC			

[Input type] Setting input

[Data type] Bit path

#3 PLC When the remaining time of an item falls to a value less than the percentage of the life specified in parameter No.8911, the life warning status is displayed at

0: Timer area on CNC status display area.

1: Alarm area on CNC status display area.

	#7	#6	#5	#4	#3	#2	#1	#0
8901	MEN							

[Input type] Setting input

[Data type] Bit path

#7 MEN The periodic maintenance screen and the maintenance information screen are:

0: Displayed.

1: Not displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
8903								PDM

[Input type] Parameter input

[Data type] Bit path

#0 PDM When the remaining time of an item falls to a value less than the percentage of the life specified in parameter No.8911, the life warning status on CNC status display area is

0: Not displayed.

1: Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
8906		MPM						

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#6 MPM On the periodic maintenance screen,
0: the life time is counted in each path.
1: the life time is not counted in each path.

8911	Percentage for life warning display on the periodic maintenance screen

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 99

On the periodic maintenance screen, if the remaining time of an item falls to a value less than the percentage of the life specified in this parameter, the item name and remaining time is displayed in red as a warning.

13.1.20 Selection of display axis on the Current Position Screen

Overview

On the current position screen, it can be selected whether to display each axis position by setting the parameter or signal.
Moreover, the order of displaying axis can be changed.

Explanation

- Selection of display axis

If bit 0 (NDPx) of parameter No. 3115 is set to 1, the set axis is not displayed and the blank is displayed.

- Selection of display the absolute and relative coordinates and the distance-to-go

If bit 1 (NDAx) of parameter No. 3115 is set to 1, the absolute and relative coordinates and the distance-to-go is not displayed. (The machine coordinates are displayed)

- Change the order of displaying axis

If parameter No. 3130 is set, the order of displaying each axis can be changed.

> **NOTE**
> 1 The order can be changed only in the current position screen, and it is impossible in other screens such as the work-offset screen and the parameter screen.
> 2 If the parameters of all axes are set to 0, this function is invalid.
> 3 The axis that 0 or value more than number of maximum axes in the path is set to the parameter is not displayed.

- Close up display

If bit 0 (TAD) of parameter No. 13102 is set to 1, when the axis is not displayed by the bit 0 (NDPx) of parameter No. 3115, the axes are closed up and displayed.
Moreover, when the order of displaying axes are changed by parameter No. 3130, the axis that 0 is set to the parameter is closed up and displayed.

- Selection of display axis by the signal

If the axis non-displayed signal NPOS1 to NPOS8 <Gn198> is set to “1”, the set axis is not displayed and the blank is displayed.

NOTE
If bit 0 (TAD) of parameter No. 13102 is set to 1, when the axis non-displayed signal NPOS1 to NPOS8 <Gn198> is set to “1”, the axis is closed up.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3115							NDAx	NDPx

[Input type] Parameter input
[Data type] Bit axis

#0 NDPx The current position is:
0: Displayed.
1: Not displayed.

NOTE
When using the electronic gear box (EGB) function, set 1 for the EGB dummy axis to disable current position display.

#1 NDAx The current position and the amount of the movement to be made in absolute and relative coordinates are:
0: Displayed.
1: Not displayed. (The machine coordinates are displayed.)

3130	Axis display order for current position display screens

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 32
Set the order in which axes are displayed on current position display screens.

	#7	#6	#5	#4	#3	#2	#1	#0
13102								TAD

[Input type] Parameter input
[Data type] Bit path

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 TAD When the axis is not displayed by the bit 0 (NDPx) of parameter or parameter No.3130,
0: The set axis is displayed as blanks.
1: The axis is closed up and displayed.

Signal

Axis non-displayed signals NPOS1 to NPOS8<Gn198>

[Classification] Input signal
[Function] The display axis on the current position screen is selected.
[Operation] When these signals are set to “0”, the set axes are displayed.
When these signals are set to “1”, the set axes are not displayed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn198	NPOS8	NPOS7	NPOS6	NPOS5	NPOS4	NPOS3	NPOS2	NPOS1

13.1.21 Power Consumption Monitor

Function Overview

The electric power consumption and integral power consumption of servo motor and spindle motor can be read through the interface of PMC window, FOCAS2 interface or etc. Custom application can observe consumed power and regenerative power of servo and spindle axes. Then you can use them to appeal your machine with the energy saving ability. Moreover, these values can be confirmed on the CNC screen.
Because the power consumption data is in the diagnosis data, it is possible to read it from user's application in the following way. For the details, please refer to each manual.

- PMC
  WINDR function code 33 Reading diagnosis data (Low-speed)
  For details of the WINDR instruction, please refer to PMC programming manual (B-64513EN).

- FOCAS2 function
  cnc_diagnoss Read diagnosis data
  cnc_diagnosr Read diagnosis data(area specified)
  For details of the function specification, please refer to a data window library specifications bundled to FOCAS1/2 Library disk (A02B-0207-K737).

- C language executor
  cnc_diagnoss Read diagnostics data
  For details of the function specification, please refer to C Language Executor programming manual (B-63943EN-3).

Displaying power consumption

Power consumption monitoring screen

The electric power data of servo axis and spindle axis consumption and regeneration can be displayed.

Fig.13.1.21 (a) Power consumption monitoring screen for 10.4" display unit

Refer to Operator's Manual (Common to Lathe System/Machining Center System) (B-64694EN) for detailed operations on Power consumption monitoring screen.

Bar-graph display of whole power consumption

The bar-graph where the entire power consumption can be confirmed is displayed in the warning message area. Power consumption can be confirmed on every screen.

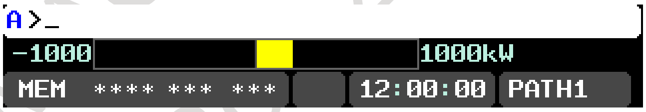


Fig.13.1.21 (b) Bar-graph of entire power consumption

NOTE

1 The bar-graph display can be concealed by the setting of parameter APM (No.11368#6)=1.

2 When the warning message is displayed, the bar-graph is not displayed.

3 The bar-graph displayed in the warning display area is only the entire power consumption.

Diagnosis data

Present power consumption of all axes

4900	Present power consumption of all axes

[Data type] 2-Word
[Unit of data] W
[Valid data range] -99999999 to 99999999

Present power consumption of each servo axis

4901	Present power consumption of each servo axis

[Data type] 2-Word axis
[Unit of data] W
[Valid data range] -99999999 to 99999999

> **NOTE**
> This value becomes negative during regeneration of power that is generated by deceleration of an axis

Present power consumption of each spindle axis

4901	Present power consumption of each spindle axis

[Data type] 2-Word spindle
[Unit of data] W
[Valid data range] -99999999 to 99999999

> **NOTE**
> This value becomes negative during regeneration of power that is generated by deceleration of an axis

Integral power consumption of all axes

4910	Integral net power consumption of all axes

4911	Integral power consumption of all axes

4912	Integral power regeneration of all axes

[Data type] 2-Word
[Unit of data] 0.001kWh
[Valid data range] -99999999 to 99999999

NOTE
- These values are integrated from 0.000kWh every turning on CNC.
- No.4911 and No.4912 are counted as “consumption” when the summational power of all axes is positive value and counted as “regeneration” when the summational power of all axes is negative value. This shows the result of including countervailing among all axes in case that some axes consume power but other axes regenerate power. Therefore these diagnosis data mean the net consumption and net regeneration power exchanging between the power supply and commercial power source.
  Example)
  Assume that the case of power
  in a spindle axis: consumption 100kWh, regeneration 25kWh
  in a servo axis: consumption 20kWh, regeneration 1kWh
  The result of simple summation becomes consumption 120kWh and regeneration 26kWh.
  In this situation, assume that 1kWh power consumption in the spindle axis and 1kWh power regeneration in the servo axis occurs simultaneously. The each 1kWh is countervailed. As a result, integral power consumption becomes 119kWh and the integral power regeneration becomes 25kWh and the integral net power consumption becomes 94kWh.

Integral power consumption of each servo axis

4920	Integral net power consumption of each servo axis
4921	Integral power consumption of each servo axis
4922	Integral power regeneration of each servo axis

[Data type] 2-Word axis
[Unit of data] 0.001kWh
[Valid data range] -99999999 to 99999999

NOTE
These values are integrated from 0.000kWh every turning on CNC.

Integral power consumption of each spindle axis

4930	integral net power consumption of each spindle axis
4931	integral power consumption of each spindle axis
4932	integral power regeneration of each spindle axis

[Data type] 2-Word spindle
[Unit of data] 0.001kWh
[Valid data range] -99999999 to 99999999

NOTE
These values are integrated from 0.000kWh every turning on CNC.

Parameter

		#7	#6	#5	#4	#3	#2	#1	#0
2281							RDPRR	RDPMU2	RDPMU1

[Input type] Parameter input
[Data type] Bit axis

#0 RDPMU1
#1 RDPMU2 Internal unit setting of power consumption monitor
Please set these bits according to the maximum motor output.

Table 13.1.21 (a) Internal unit setting (Servo)

Maximum output of motor	RDPMU2	RDPMU1
300kW or more	0	0
30kWto 300kW	0	1
3kW to 30kW	1	0
3kW or less	1	1

Table 13.1.21 (b) Setting for each servo motor

	Servo Motor		RDPMU2	RDPMU1
α*i*S500/3000HV [*2]	α*i*S1000/2000HV [*2] α*i*S1000/3000HV [*2]	α*i*S2000/2000HV [*1,2] α*i*S3000/2000HV [*1,2]	0	0
α*i*F40/3000 α*i*F40/3000FAN α*i*S12/6000 α*i*S12/6000HV α*i*S22/4000 α*i*S22/4000HV α*i*S22/6000 α*i*S22/6000HV α*i*S30/4000 α*i*S30/4000HV α*i*S40/4000	α*i*S40/4000HV α*i*S50/2000 α*i*S50/2000HV α*i*S50/3000FAN α*i*S50/3000HVFAN α*i*S60/2000 α*i*S60/2000HV α*i*S60/3000FAN α*i*S60/3000HVFAN α*i*S100/2500 α*i*S100/2500FAN	α*i*S100/2500HV α*i*S100/2500HVFAN α*i*S200/2500 α*i*S200/2500FAN α*i*S200/2500HV α*i*S200/2500HVFAN α*i*S300/2000 [*2] α*i*S300/2000HV [*2] α*i*S300/3000HV α*i*S500/2000 [*2] α*i*S500/2000HV [*2]	0	1
α*i*F2/5000 α*i*F4/4000 α*i*F4/4000HV α*i*F8/3000 α*i*F8/3000HV α*i*F12/3000 α*i*F12/3000HV α*i*F22/3000 α*i*F22/3000HV α*i*F30/3000 α*i*S2/5000 α*i*S2/5000HV α*i*S2/6000 α*i*S2/6000HV	α*i*S4/5000 α*i*S4/5000HV α*i*S4/6000 α*i*S4/6000HV α*i*S8/6000 α*i*S8/6000HV α*i*S8/4000 α*i*S8/4000HV α*i*S12/4000 α*i*S12/4000HV β*i*S1/6000 β*i*S4/4000 β*i*S4/4000HV β*i*Sc4/4000	β*i*S8/3000 β*i*S8/3000HV β*i*Sc8/3000 β*i*S12/2000 β*i*S12/3000 β*i*S12/3000HV β*i*Sc12/3000 β*i*S22/2000 β*i*S22/2000HV β*i*S22/3000 β*i*S22/3000HV β*i*S30/2000 β*i*S30/2000HV β*i*S40/2000 β*i*S40/2000HV	1	0
α*i*F1/5000 β*i*S0.2/5000 β*i*S0.3/5000	β*i*S0.4/5000 β*i*S0.5/6000	β*i*S2/4000 β*i*S2/4000HV β*i*Sc2/4000	1	1

Synchronous built-in servo motor [*3]			RDPMU2	RDPMU1
			0	0
D*i*S1500/250(400V)	D*i*S2000/150(400V)	D*i*S5000/50(400V)	0	1
D*i*S15/1000(400V) D*i*S60/400(400V) D*i*S70/300(400V) D*i*S85/1000(200V) D*i*S110/1000(200V) D*i*S150/300(200V) D*i*S150/300(400V) D*i*S200/300(200V) D*i*S200/300(400V) D*i*S250/250(200V) D*i*S250/250(400V)	D*i*S260/300(400V) D*i*S260/1000(200V) D*i*S370/300(200V) D*i*S370/300(400V) D*i*S400/250(200V) D*i*S400/250(400V) D*i*S500/250(200V) D*i*S500/250(400V) D*i*S800/250(200V) D*i*S800/250(400V)	D*i*S1000/200(200V) D*i*S1000/200(400V) D*i*S1200/250(200V) D*i*S1200/250(400V) D*i*S1500/100(200V) D*i*S1500/100(400V) D*i*S1500/250(200V) D*i*S2000/100(200V) D*i*S2000/100(400V) D*i*S2000/150(200V)	1	0
D*i*S15/1000(200V) D*i*S22/600(200V) D*i*S22/600(400V) D*i*S22/1500(200V)	D*i*S60/400(200V) D*i*S70/300(200V) D*i*S85/400(200V) D*i*S85/400(400V)	D*i*S110/400(200V) D*i*S110/400(400V) D*i*S260/300(200V)	1	1

Linear motor [*4]			RDPMU2	RDPMU1
			0	0
L*i*S17000C3/2(400V)			0	1
L*i*S1500B1/4(200V) L*i*S3000B2/2(200V) L*i*S3000B2/2(400V) L*i*S3000B2/4(200V) L*i*S4500B2/2(200V) L*i*S4500B2/2(400V) L*i*S4500B2/2HV L*i*S6000B2/2(200V) L*i*S6000B2/2(400V) L*i*S6000B2/2HV L*i*S6000B2/4(200V)	L*i*S7500B2/2(200V) L*i*S7500B2/2(400V) L*i*S7500B2/2HV L*i*S9000B2/2(200V) L*i*S9000B2/2(400V) L*i*S9000B2/4(200V) L*i*S3300C1/2(200V) L*i*S3300C1/2(400V) L*i*S9000C2/2(200V) L*i*S9000C2/2(400V) L*i*S9000C2/2HV	L*i*S11000C2/2(200V) L*i*S11000C2/2(400V) L*i*S11000C2/2HV L*i*S15000C2/2(200V) L*i*S15000C2/2(400V) L*i*S15000C2/3(200V) L*i*S15000C2/3HV L*i*S10000C3/2(200V) L*i*S10000C3/2(400V) L*i*S10000C3/2HV L*i*S17000C3/2(200V) L*i*S17000C3/2HV	1	0
L*i*S300A1/4(200V)	L*i*S600A1/4(200V)	L*i*S900A1/4(200V)	1	1

*1) Power Consumption Monitor is not available to the axis using α*i*S3000/2000HV or α*i*S2000/2000HV with torque control.
*2) In the case of “plural winding motor”, which is a motor driven by 2 or 4 amplifiers, please set the following parameters for main axis only. (Please set to 0 for sub axis)
*3) In the case of synchronous built-in servo motors, please set No.2300#2=1. If you use a non-binary type encoder, please set No.2010#2=1 at the same time.
*4) In the case of linear motors, please set No.2010#2=1.
*5) Regarding the other motors, please set it according to the maximum motor output described in each motor manual.

> **NOTE**
> If these bits are wrong, the electric power consumption data is not calculated correctly.

#2 RDPRR Regeneration method of servo amplifier is
0: regeneration to power source
1: register regeneration

> **NOTE**
> In case of using the amplifier with register regeneration, regeneration power is not returned to power source. Therefore, instantaneous power consumption should become greater than 0.

2468	RDPMR Motor winding resistance

[Input type] Parameter input
[Data type] Word axis
[Unit of data] $2^{-12}\,\Omega$

Please set the value of "Armature resistance (1 phase)" in the specifications of the motor.
Example) Case of αi S8/4000

Set value=$0.31/2^{-12}$=1270 where armature resistance=0.31 Ω

Data sheet

Parameter	Symbol	Value	Unit
Stall Torque (*)	Ts	8.0 82	Nm kgfcm
Stall Current (*)	Is	11.1	A (rms)
Rated Output (*)	Pr	2.5 3.3	kW HP
Rating Speed	Nr	4000	min^{-1}
Maximum Speed	Nmax	4000	min^{-1}
Maximum Torque (*)	Tmax	32 327	Nm kgfcm
Rotor Inertia	Jm	0.00117 0.0119	kgm^2 $kgfcms^2$
Rotor Inertia (with Brake)	Jm	0.00124 0.0127	kgm^2 $kgfcms^2$
Torque constant (*)	Kt	0.72 7.3	Nm/A (rms) kgfcm/A (rms)
Back EMF constant (1phase) (*)	Ke Kv	25 0.24	V (rms)/1000 min^{-1} V (rms)sec/rad
Armature Resistance (1 phase) (*)	Ra	0.31	Ω
Mechanical time constant	tm	0.002	s
Thermal time constant	tt	20	min
Static friction	Tf	0.3 3	Nm kgfcm
Weight	w	7.4	kg
Weight (with Brake)	w	9.6	kg
Max. Current of Servo Amp.	Imax	80	A (peak)

> **NOTE**
> 1 Be sure to set this parameter. Power consumption calculation works when this parameter is set to more than 1.
> 2 Power consumption calculation works on main axis only in case of the motor with plural winding. Set this parameter for a main axis only. Set 0 for sub axis.

2469	RDPMA Lose coefficient A of a servo amplifier

[Input type] Parameter input
[Data type] Word axis

Please set the value of loss coefficient A in Table Table 13.1.21 (c) .
Example) Case of αi SV80-B, Set value=832

Case of αi SV80/80-B, Set value (L axis) = 672, Set value (M axis) = 672

> **NOTE**
> 1 Set this value only to main axis in case of a plural winding motor. Set 0 to the sub axis.
> 2 When you use Serial feedback dummy function with servo amplifier, add RDPMA value of dummy axis to No.2469 of other non-dummy axis.

2490	RDPMB Lose coefficient B of a servo amplifier

[Input type] Parameter input
[Data type] Word axis

Please set the value of loss coefficient B in Table Table 13.1.21 (c) .
Example) Case of α*i* SV80-B, α*i* SV80/80-B HRV2, Set value=262
Case of α*i* SV80-B, α*i* SV80/80-B HRV3, Set value=352

> **NOTE**
> 1 Setting value depends on HRV.
> 2 Set this value only to main axis in case of a plural winding motor. Set 0 to the sub axis.

Table 13.1.21 (c) Loss coefficient A, B setting value of servo amplifier

In case of multi-axis amplifier, please set the value corresponding to each axis. If there are two columns, the upper column corresponds to L axis and the lower correspond to M axis. If there are three columns, each corresponds to L, M and N axis from the upper.

Name	Spec. (A06B)	Loss coefficient A	Loss coefficient B	
			HRV2	HRV3
α*i*SV4-B	6240-H101	832	307	397
α*i* SV20-B	6240-H103	832	307	397
α*i* SV40-B	6240-H104	832	269	339
α*i* SV80-B	6240-H105	832	262	352
α*i* SV160-B	6240-H106	1152	288	371
α*i* SV360-B	6240-H109	1600	301	352
α*i* SV4/4-B	6240-H201	608	307	397
		608	307	397
α*i* SV4/20-B	6240-H203	608	307	397
		608	307	397
α*i* SV20/20-B	6240-H205	608	307	397
		608	307	397
α*i* SV20/40-B	6240-H206	608	307	397
		608	269	339
α*i* SV40/40-B	6240-H207	672	269	339
		672	269	339
α*i* SV40/80-B	6240-H208	672	269	339
		672	262	352
α*i* SV80/80-B	6240-H209	672	262	352
		672	262	352
α*i* SV80/160-B	6240-H210	736	262	352
		736	288	371
α*i* SV160/160-B	6240-H211	736	288	371
		736	288	371
α*i* SV4/4/4-B	6240-H301	555	307	397
		555	307	397
		555	307	397

Name	Spec. (A06B)	Loss coefficient A	Loss coefficient B	
			HRV2	HRV3
αi SV20/20/20-B	6240-H305	555	307	397
		555	307	397
		555	307	397
αi SV20/20/40-B	6240-H306	555	307	397
		555	307	397
		555	269	339
αi SV40/40/40-B	6240-H308	661	269	339
		661	269	339
		661	269	339
αi SVP20/20/20-5.5-B	6230-H001	704	307	397
		704	307	397
		704	307	397

Name	Spec. (A06B)	Loss coefficient A	Loss coefficient B	
			HRV2	HRV3
αi SV10HV-B	6290-H102	832	499	902
αi SV20HV-B	6290-H103	832	538	883
αi SV40HV-B	6290-H104	832	538	928
αi SV80HV-B	6290-H105	1152	550	749
αi SV180HV-B	6290-H106	1600	538	749
αi SV360HV-B	6290-H109	2176	474	672
αi SV540HV-B	6290-H110	2752	512	-
αi SV10/10HV-B	6290-H202	608	499	902
		608	499	902
αi SV10/20HV-B	6290-H204	608	499	902
		608	538	883
αi SV20/20HV-B	6290-H205	672	538	883
		672	538	883
αi SV20/40HV-B	6290-H206	672	538	883
		672	538	928
αi SV40/40HV-B	6290-H207	672	538	928
		672	538	928
αi SV40/80HV-B	6290-H208	736	538	928
		736	550	749
αi SV80/80HV-B	6290-H209	736	550	749
		736	550	749
αi SV10/10/10HV-B	6290-H302	555	499	902
		555	499	902
		555	499	902
αi SV10/10/20HV-B	6290-H303	555	499	902
		555	499	902
		555	538	883
αi SV20/20/20HV-B	6290-H305	661	538	883
		661	538	883
		661	538	883
αi SV10/10/10-5.5HV-B	6280-H001	1056	499	902
		1056	499	902
		1056	499	902

Note) If β*i*SV-B amplifiers, which adopt resistor regeneration, are used, please set No2281#2=1.

Name	Spec. (A06B)	Loss coefficient A	Loss coefficient B	
			HRV2	HRV3
β*i*SV4-B	6160-H001	960	320	416
β*i*SV20-B	6160-H002	960	320	416
β*i*SV40-B	6160-H003	960	294	378
β*i*SV80-B	6160-H004	960	275	371
β*i*SV20/20-B	6166-H201	512	320	416
		512	320	416
β*i*SV40/40-B	6166-H203	512	288	365
		512	288	365
β*i*SV10HV-B	6161-H001	960	525	947
β*i*SV20HV-B	6161-H002	960	563	928
β*i*SV40HV-B	6161-H003	960	563	979

2463	RDPMC Loss coefficient C of power supply

[Input type] Parameter input
[Data type] Word axis

Please set the value of loss coefficient C in Table 13.1.21 (d) to the PS loss calculating axis. Please set 0 to the other axis.
PS lose calculating axis is :
One of the axes whose amplifier is connected to a common power supply (PS).
On PS loss calculating axis, the power loss of power supply (PS) is added to the power consumption of the axis.

NOTE
If both main axis and sub axis of a plural winding motor have PS respectively, set double value as loss coefficient C for main axis, and set 0 for sub axis.

2491	RDPMD Loss coefficient D of power supply

[Input type] Parameter input
[Data type] Word axis

Please set the value of loss coefficient D in Table 13.1.21 (d).

NOTE
1 Set the value to all servo axes.
2 Set this value only to main axis in case of a plural winding motor. Set 0 to the sub axis.

Table 13.1.21 (d) Loss coefficient C, D setting of power supply

Name	Spec. (A06B)	C	D
α*i* PS3-B	6200-H003	883	730
α*i* PS7.5-B	6200-H008	1037	659
α*i* PS11-B	6200-H011	1037	659
α*i* PS15-B	6200-H015	1037	653
α*i* PS26-B	6200-H026	1344	717
α*i* PS30-B	6200-H030	1344	704
α*i* PS37-B	6200-H037	1344	723
α*i* PS55-B	6200-H055	2266	672
α*i* PS11HV-B	6250-H011	1120	397
α*i* PS18HV-B	6250-H018	1120	378
α*i* PS30HV-B	6250-H030	1427	499
α*i* PS45HV-B	6250-H045	1427	486
α*i* PS60HV-B	6250-H060	1427	486
α*i* PS75HV-B	6250-H075	2349	480
α*i* PS100HV-B	6250-H100	2349	480

If β*i*SV-B amplifiers, which adopt resistor regeneration, are used, please set No2281#2=1.
*1) The value in case of single-phase input.

Name	Spec. (A06B)	C	D
β*i*SV4-B	6160-H001	0	621 (1024) *1
β*i*SV20-B	6160-H002	0	621 (1024) *1
β*i*SV40-B	6160-H003	0	646
β*i*SV80-B	6160-H004	0	646
β*i*SV20/20-B	6166-H201	0	710
β*i*SV40/40-B	6166-H203	0	710
β*i*SV10HV-B	6161-H001	0	320
β*i*SV20HV-B	6161-H002	0	320
β*i*SV40HV-B	6161-H003	0	320

2105	TRQCST Torque constant number

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.00001Nm / 1 Torque command
[Valid data range] 1 to 32767

This parameter is used to specify a motor-specific torque constant.
Please check this parameter is same to standard setting value according to motor model.

	#7	#6	#5	#4	#3	#2	#1	#0
2301	TQCT10							

[Input type]Parameter input
[Data type] Bit axis

#7 TQCT10 Setting of torque constant (No.2105) is used
0: without being any modified.
1: after being multiplied by 10.
Please check this parameter is same to standard setting value according to motor model.

	#7	#6	#5	#4	#3	#2	#1	#0
2200						IQOB		

[Input type]Parameter input
[Data type] Bit axis

#2 IQOB IQ observer is
0: invalid.
1: valid.
Please set 1.

	#7	#6	#5	#4	#3	#2	#1	#0
4541						PWCBT1	PWCBT0	POWCAL

[Input type]Parameter input
[Data type] Bit spindle

#0 POWCAL Power consumption calculation of spindle is
0: invalid.
1: valid.

#1 PWCBT0
#2 PWCBT1 Internal unit setting of power consumption monitor.
Please set these bits according to the maximum motor output.

Table 13.1.21 (e) Internal unit setting (Spindle)

Maximum output of motor	PWCBT1	PWCBT0
300kW or more	0	0
30kWto 300kW	0	1
3kWto 30kW	1	0
3kW or less	1	1

Table 13.1.21 (f) Setting for each spindle motor

	Spindle motor		PWCBT1	PWCBT0
αiI 22/8000-B αiI 40/7000-B αiIP 60/5000-B αiI 40/7000HV-B αiIP 50/6000HV-B	αiI 22/12000-B αiI 50/5000-B αiI 22/8000HV-B αiI 60/5000HV-B αiIP 60/5000HV-B	αiI 30/7000-B αiIP 50/6000-B αiI 30/7000HV-B αiI 100/5000HV-B	0	1
αiI 1/15000-B αiI 2/10000-B αiI 3/12000-B αiI 8/8000-B αiIP 12/8000-B αiIP 22/8000-B αiI 2/10000HV-B αiI 8/80000HV-B αiIP 22/8000HV-B	αiI 1.5/10000-B αiI 2/20000-B αiI 6/10000-B αiI 8/10000-B αiIP 15/8000-B αiIP 30/6000-B αiI 3/10000HV-B αiI 12/8000HV-B αiIP 40/6000HV-B	αiI 1.5/15000-B αiI 3/10000-B αiI 6/12000-B αiI 12/8000-B αiIP 18/8000-B αiI 1.5/10000HV-B αiI 6/10000HV-B αiIP 15/8000HV-B	1	0
αiI 0.5/10000-B	αiI 0.5/10000HV-B	αiI 1/10000HV-B	1	1

Regarding the other motors, please set the value according to the maximum motor output described in each motor manual.

4593	Loss coefficient E of a spindle amplifier

[Input type] Parameter input
[Data type] Word spindle
Please set the value of loss coefficient E in Table 13.1.21 (g).

4594	Loss coefficient F of a spindle amplifier

[Input type] Parameter input
[Data type] Word spindle
Please set the value of loss coefficient F in Table 13.1.21 (g).

Table 13.1.21 (g) Loss coefficient E and F of spindle amplifier

Spindle amplifier	Spec. (A06B)	Loss coefficient E	Loss coefficient F
αi SP2.2-B	6220-H002	1043	326
αi SP5.5-B	6220-H006	1158	371
αi SP11-B	6220-H011	1274	326
αi SP15-B	6220-H015	1274	320
αi SP22-B	6220-H022	1478	378
αi SP26-B	6220-H026	1478	346
αi SP30-B	6220-H030	1478	346
αi SP37-B	6220-H037	1478	352
αi SP45-B	6220-H045	2662	326
αi SP55-B	6220-H055	2662	346
αi SVP20/20/20-5.5-B	6230-H001	0	371
αi SP5.5HV-B	6270-H006	1523	659
αi SP11HV-B	6270-H011	1523	653
αi SP15HV-B	6270-H015	1523	608
αi SP22HV-B	6270-H022	1766	704
αi SP30HV-B	6270-H030	1766	704
αi SP45HV-B	6270-H045	1766	730
αi SP75HV-B	6270-H075	2656	723
αi SP100HV-B	6270-H100	2656	723
αi SVP10/10/10-5.5HV-B	6280-H001	0	659

Please set above coefficient regardless of PWM frequency setting.

4595	Loss coefficient C of a power supply

[Input type] Parameter input
[Data type] Word spindle
Please set the value of loss coefficient C in Table 13.1.21 (d) to the PS loss calculating axis. Please set 0 to the other axis.
PS lose calculating axis is :
One of the axes whose amplifier is connected to a common power supply (PS).
On PS loss calculating axis, the power loss of power supply (PS) is added to the power consumption of the axis.

4596	Loss coefficient D of a power supply

[Input type] Parameter input
[Data type] Word spindle
Please set the value of loss coefficient D in Table 13.1.21 (d).

> **NOTE**
> 1 Set the value to all spindle axes.
> 2 Even if you used plural winding spindle motor, set the specified value in Table 13.1.21 (d) as loss coefficient D in respective axis.

	#7	#6	#5	#4	#3	#2	#1	#0
11368		APM	PWC					

[Input type] Parameter input
[Data type] Bit

#5 PWC Power consumption monitoring screen is
0: invalid.
1: valid.

#6 APM Bar-graph display that shows the whole of power consumption is
0: valid.
1: invalid.

> **NOTE**
> This parameter is effective at PWC=1.

11371	Scale for PCM bar-graph

[Input type] Parameter input
[Data type] Word
[Unit of data] kW
[Valid data range] 0 to 9999

Specify the absolute value of maximum/minimum scale for the bar graph of power consumption monitoring screen function.
If 0 is specified, the setting of parameter No.2281#0, #1(servo), and parameter No.4541#1 and #2(spindle) are checked, and the biggest motor output value is used as a maximum/minimum value for scale.
e.g.) If 3000 is specified, -3000kW/3000kW are displayed.

11392	Scale for PCM bar-graph of servo axis

[Input type] Parameter input
[Data type] 2-Word axis
[Unit of data] 0.001kW
[Valid data range] 0 to 99999999

Specify the absolute value of maximum/minimum scale for the bar graph of each servo axis in power consumption monitoring screen.
Specify integer from 1 to 9999999.
4 digits from the top are displayed in power consumption monitoring screen, though the value of bar graph is estimated by using this parameter itself.
If 0 is specified, the scale is decided by the parameters 2281#0 and #1.
e.g.) If 9999999 is specified, -9999kW/9999kW are displayed.
If 11100 is specified, -11.10kW/11.10kW are displayed.

11393	Scale of PCM bar-graph of spindle axis

[Input type] Parameter input
[Data type] 2-Word spindle
[Unit of data] 0.001kW
[Valid data range] 0 to 99999999

Specify the absolute value of maximum/minimum scale for the bar graph of each spindle axis in power consumption monitoring screen.
Specify integer from 1 to 9999999.
4 digits from the top are displayed in power consumption monitoring screen, though the value of bar graph is estimated by using this parameter itself.
If 0 is specified, the scale is decided by the parameters 4541#1 and #2.
e.g.) If 9999999 is specified, -9999kW/9999kW are displayed.
If 11100 is specified, -11.10kW/11.10kW are displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
11394								AND

[Input type]Parameter input
[Data type] Bit axis

#0 AND Power consumption of each servo axis is
0: displayed.
1: not displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
11395								SND

[Input type]Parameter input
[Data type] Bit spindle

#0 SND Power consumption of each spindle axis is
0: displayed.
1: not displayed.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Power consumption monitoring screen

13.1.22 Energy Saving Level Selecting Function

Function Overview

Power consumption in a machine can be categorized into power consumption in spindle motors, that in servo motors and that in peripheral devices (such as a coolant pump) (See Fig. 13.1.22 (a)). Suppressing torque at acceleration/deceleration in spindles, where there is large power consumption, can reduce whole power consumption of a machine, though machining time gets longer.
By utilizing this feature, this function makes it possible to switch between machining with shorter time and that with less power consumption.

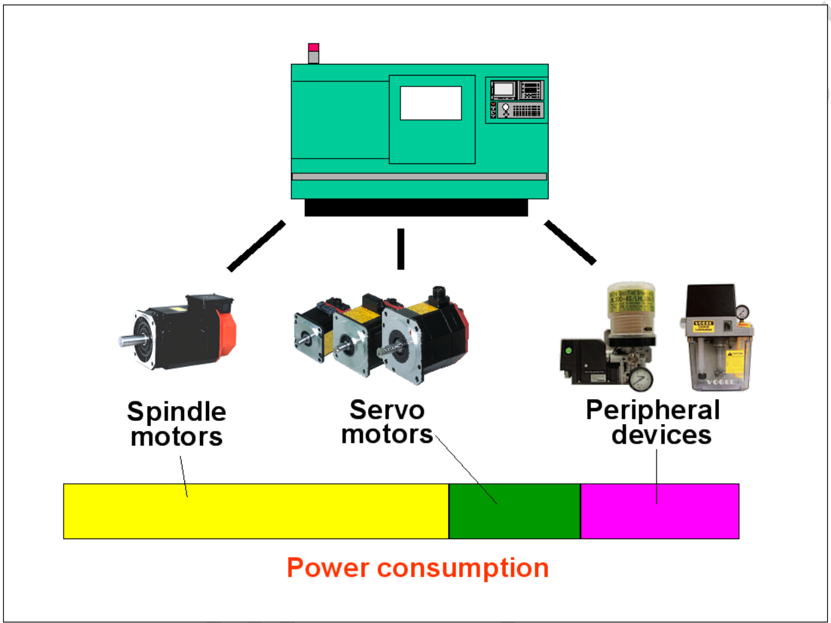


Fig. 13.1.22 (a) Power consumption in machine

Fig. 13.1.22 (b) shows the relation between power consumption and machining time.
Power consumption in spindle motors gets smaller by suppressing spindles' torque at acceleration/deceleration. However, machining time gets longer because acceleration/deceleration requires longer time. Therefore, the longer machining time gets, the smaller the summation of power consumption in spindle motors and in servo motors gets as shown in Fig. 13.1.22 (b). On the other hand, longer machining time makes power consumption in peripheral devices larger because running times of them get longer (See power consumption in peripheral devices in Fig. 13.1.22 (b)). Total power consumption is the sum of these power consumptions above.
'A' in Fig. 13.1.22 (b) shows the point of the fastest machining (normal machining). Utilizing full spindles' torque makes required time for acceleration/deceleration smaller, though total power consumption gets larger because of large power consumption in spindle motors.
'B' shows the point of the least power consumption. As described above, machining time gets longer because of suppression in spindles' torque at acceleration/deceleration.

This function makes it possible to select machining levels (called Eco levels) between the fastest machining ('A') and machining with the least power consumption ('B').

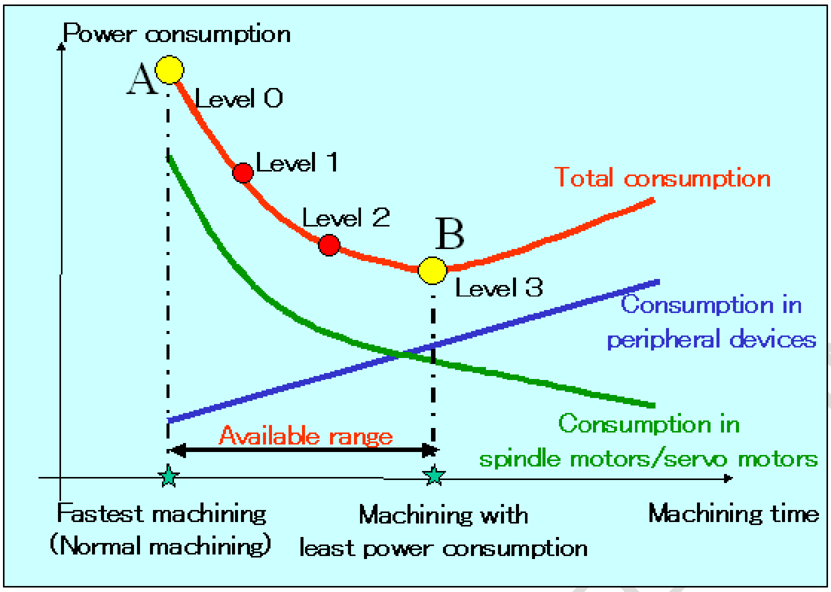


Fig. 13.1.22 (b) Power consumption vs. machining time

Therefore, selecting an Eco level provides both faster machining and less power machining.
By using this function, power consumption can be reduced effectively when, for example, there is enough time to deadline or there is large difference in production time of each part in production line.

NOTE

1. This function can reduce power consumption in spindles only at acceleration/deceleration. Power consumption during cutting does not change even if the selected Eco level changes. Therefore, the effect of eco-machining may get smaller for machining with less acceleration/deceleration of spindles.
2. Power consumption of a spindle motor may fluctuate depending on the conditions of the motor temperature and the power supply voltage even though the motor runs in the same operation.

Setting Overview

Three settings listed below are required before using this function.

- Setting to obtain power consumption in spindle and servo motors
- Setting to obtain power consumption in peripheral devices
- Setting to decide the point of the least power consumption

Obtaining power consumption in spindle and servo motors

Since power consumption in spindle and servo motors referred in this function is obtained by Power consumption monitor, settings for the function are required.
Refer to Operator's Manual (Common to Lathe System/Machining Center System) (B-64694EN) for detail.

Obtaining power consumption in peripheral devices

Some settings are needed to obtain power consumption in peripheral devices such as oil pressure pumps, illuminations and so on.
Power consumption in peripheral devices can be calculated by average power and operating time of each device.
These settings are performed on Power consumption setting screen.
Refer to Operator's Manual (Common to Lathe System/Machining Center System) (B-64694EN) for detailed operations on the screen.

Deciding the point of the least power consumption

Adjustment of spindles' torque is required so that Eco level 0 means normal machining and Eco level 3 means machining with the least power consumption as shown in Fig. 13.1.22 (b).
No adjustment is needed for Eco level 0 because this is normal setting.
Adjust suppression on spindles' torque at acceleration/deceleration (torque override) so that the sum of power consumption in spindles, servos and peripherals is minimum at Eco level 3.
Eco levels 1 and 2 are automatically decided as settings dividing between Eco level 0 and 3 equally.
Refer to below for detailed explanations on torque overrides.

> **NOTE**
> 1 In spindle speed control mode, the torque command is limited at acceleration/deceleration according to the setting of torque overrides. In spindle synchronization control mode and in rigid tapping mode, the time constants for acceleration/deceleration are set longer according to the torque overrides.
> 2 If more detailed selection of an Eco level is needed, set bit 0 (ELV) of the parameter No. 24303 to 1, which enhances divisions of Eco levels from 4 to 8.

About torque override

Torque override and output

Spindles' torque at acceleration/deceleration used in eco-machining is that in normal machining multiplied by torque override.
When torque override is set to 100%, torque in eco-machining is the same as that in normal machining.
If torque override is set to 50%, the torque becomes half.

Sorts of torque overrides

There are three torque overrides; one is for spindle speed control mode, one is for spindle synchronization control mode and the other is for rigid tapping mode.

Relation between Torque overrides and Eco levels

The relation between torque overrides and Eco levels is shown as follows:

- Eco level 0 means normal machining, where torque overrides are equal to 100%.
- Eco level 3 means machining with torque overrides specified by the parameters Nos. 11397, 11398 and 5260. Note that these parameters can be changed on Eco setting screen.
- Eco levels 1 and 2 are defined as machining with torque overrides dividing between Eco level 0 (torque override 100%) and 3 equally. When the parameter No.11397 is set to 70, for example, torque override of Eco level 1 is equal to 90% and that of level 2 is equal to 80%.

> **NOTE**
> 1 When the number of divisions of Eco levels is equal to 8 (bit 0 (ELV) of the parameter No.24303 is set to 1), torque overrides for level 7 are specified by the parameters Nos. 11397, 11398 and 5260. Eco levels 1 to 6 are defined as machining with torque overrides dividing between Eco level 0 and 7 equally.
> 2 Decimal sections of torque overrides are half adjusted.

Setting Details

Obtaining power consumption in peripheral devices

Power consumption in peripheral devices is calculated by average power and running time of each device. Specify average power and reference PMC address, which is used to detect ON/OFF state of each device, on Power consumption setting screen (Fig. 13.1.22 (c)).
Up to 8 devices can be recorded for each path.
Complete this setting by the examples below.
Refer to Operator’s Manual (Common to Lathe System/Machining Center System) (B-64694EN) for detailed operations on Power consumption setting screen.

> **NOTE**
> This setting is required for each path in multi-path systems.

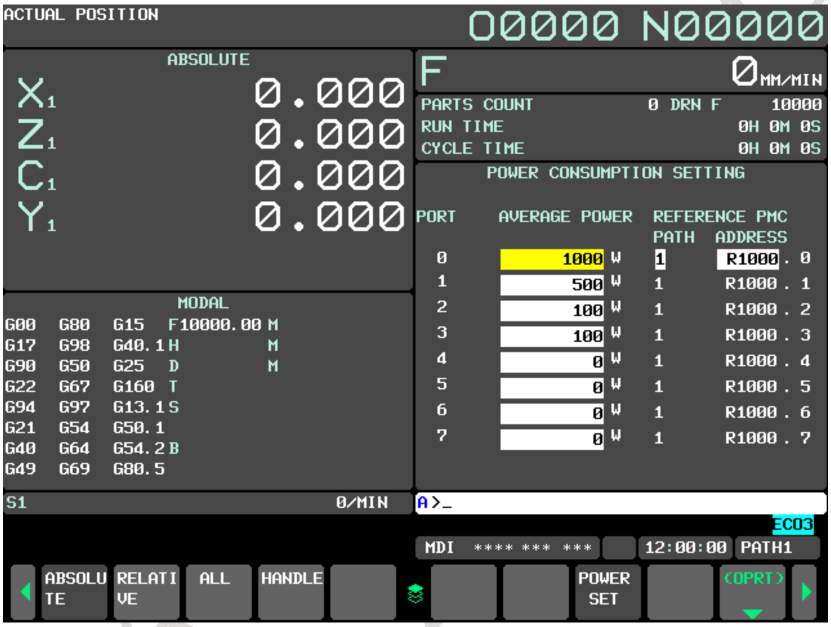
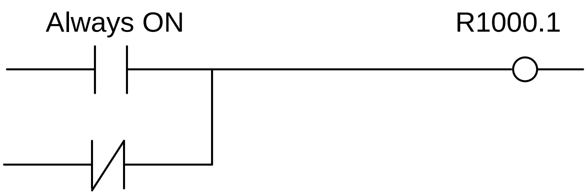


Fig. 13.1.22 (c) Power consumption setting screen

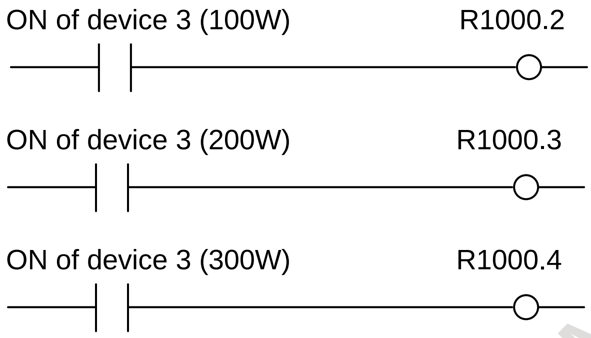
Examples

1 Complete setting by the examples below, corresponding to the ways to utilize devices.
In the case of a device switched ON/OFF during automatic operation:
Set 100W and R1000.0 into average power and PMC address of Port 0 respectively.
Use ladder programs including the ladder like below and set ON/OFF state of the device to the address.

2 In the case of a device always in operation during automatic operation:
Set 100W and R1000.1 into average power and PMC address of Port 1 respectively.
Use ladder programs including the ladder like below and set ON state of the device to the address at any time.



3 In the case of a device whose power varies during operation:
Set 100W/R1000.2 to Port 2, 200W/R1000.2 to Port 3 and 300W/R1000.4 to Port 4 respectively.
Use ladder programs including the ladders like below.
Change addresses to set ON state of the device according to power consumption of the device.



4 In the case of a device used in multiple paths:
Divide average power by the number of paths and input the calculated value for each path.
When paths 1 and 2 use the same device with average power 100W, for example, 50W is assumed to be input to Port 0 of path 1 and Port 0 of path 2. ON/OFF state of the device is needed to set into the addresses specified in Port 0 of path 1 and 2 respectively.

Deciding the point of the least power consumption

Deciding the point of the least power consumption is performed on Eco setting screen.
Decide torque override for Eco level 3 with the help of data of power consumption and cycletime obtained by actual operations so that total power consumption is minimum at Eco level 3. Adjust torque override referring to the example below.
Refer to Operator’s Manual (Common to Lathe System/Machining Center System) (B-64694EN) for detailed operations on Eco setting screen.

Example

Decide setting for the least power consumption by repeating operations at different torque overrides like below.

1 Set bit 0 (ELV) of the parameter No.24303 to 1 in order to enhance divisions of Eco levels from 4 to 8. In addition, switch the displayed unit of power consumption into “Wh” by soft-key operations. These settings make it possible to decide the point of the least power consumption more precisely, though these are not indispensable.
2 Set bit 0 (PWE) of the parameter No.8900 to 1 so that torque override for Eco level 7 can be input on Eco setting screen.
3 Set torque override for Eco level 7 to 30%.
4 Operate a test program at each Eco level 0 to 7 and obtain data of power consumption and cycletime (An example of the result is shown in Fig. 13.1.22 (d).).
A test program including more acceleration/deceleration of spindles makes it possible to decide the point of the least power consumption more precisely because torque override effects on acceleration/deceleration of spindles.
5 Assume the range of torque overrides minimizing power consumption from the graph on Eco setting screen (Fig. 13.1.22 (d) shows that torque override minimizing power consumption is between 60 and 80.).
6 Set torque override for Eco level 7 to 60%, which is the minimum value of the range assumed in 0th step above.
7 Operate the test program at each Eco level 0 to 7 again and obtain data of power consumption and cycletime (An example of the result is shown in Fig. 13.1.22 (e)).

8 Assume the range of torque overrides minimizing power consumption again (Fig. 13.1.22 (e) shows that torque override minimizing power consumption is between 66 and 77.).
9 Repeat above operations in order to assume desired torque override more precisely.
10 After finishing this adjustment, set bit 0 (ELV) of the parameter No.24303 to 0 (set the number of divisions of Eco levels to 4), if needed.

NOTE
1 When using spindle synchronization control or rigid tapping, decision of the point of the least power consumption is needed for spindle synchronization control mode or for rigid tapping mode respectively, as well as for spindle speed control mode.
2 With multiple spindles, decision of the point of the least power consumption is needed for each spindle.

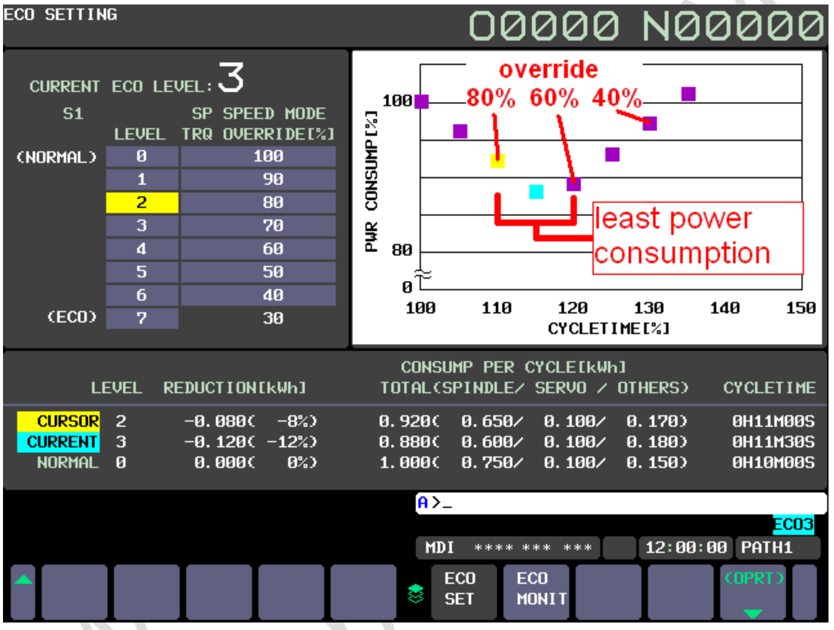


Fig. 13.1.22 (d) Deciding the point of the least power consumption 1

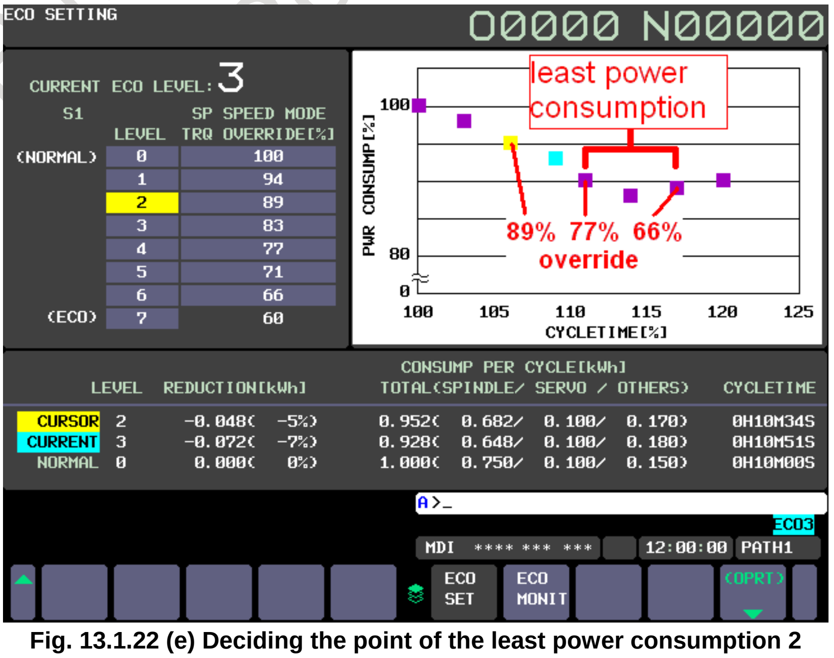


Fig. 13.1.22 (e) Deciding the point of the least power consumption 2

Restriction

This function can be used when Energy saving level selecting function is available in all spindle amplifiers.
In addition, analog spindles cannot be used in this function.

Diagnosis data

Accumulated power consumption

4923	Accumulated power consumption in servo axes

4933	Accumulated power consumption in spindle axes

4943	Accumulated power consumption in peripheral devices

4953	Accumulated reduction of power consumption

[Data type] 2-Word
[Unit of data] 0.001kWh
[Valid data range] -2147483648 to 2147483647

> **NOTE**
> These data have been accumulated since last clear operation.
> These data are not cleared by power off.

Parameter

11397	Minimum torque overrides at acceleration/deceleration in spindle speed control mode

[Input type] Parameter input
[Data type] Byte spindle
[Unit of data] %
[Valid data range] 0 to 100

Set minimum torque override at acceleration/deceleration in spindle speed control mode. This value corresponds to Eco level 3 (when bit 0 (ELV) of the parameter No.24303 is set to 0) or Eco level 7 (when bit 0 (ELV) of the parameter No.24303 is set to 1).

When torque override is set to 50%, for example, time constant for acceleration/deceleration gets twice.
Note that 0% is considered as 100%.
If eco-machining is not necessary, please set 0 or 100 in this parameter.

When bit 0 (PWE) of the parameter No.8900 is set to 1, this value can be changed on Eco setting screen.

> **NOTE**
> This parameter requires the option of Energy saving level selecting function.

11398	Minimum torque overrides at acceleration/deceleration in spindle synchronization control mode

[Input type] Parameter input
[Data type] Byte spindle
[Unit of data] %
[Valid data range] 0 to 100

Set minimum torque override at acceleration/deceleration in spindle synchronization control mode.
This value corresponds to Eco level 3 (when bit 0 (ELV) of the parameter No.24303 is set to 0) or Eco level 7 (when bit 0 (ELV) of the parameter No.24303 is set to 1).

When torque override is set to 50%, for example, time constant for acceleration/deceleration gets twice.
Note that 0% is considered as 100%.
If eco-machining is not necessary, please set 0 or 100 in this parameter.

When bit 0 (PWE) of the parameter No.8900 is set to 1, this value can be changed on Eco setting screen.

> ⚠ **CAUTION**
> Machine or work can be damaged by out-of-synchronization when torque overrides of spindles in synchronization control mode differ from each other. This is because this function differs the time constant for acceleration/deceleration in spindle synchronization mode.
> Set a common value for spindles belonging to the same group of synchronization.

> **NOTE**
> This parameter requires the option of Energy saving level selecting function.

5260	Minimum torque overrides at acceleration/deceleration in rigid tapping

[Input type] Parameter input
[Data type] Byte spindle
[Unit of data] %
[Valid data range] 0 to 100

Set minimum torque override at acceleration/deceleration in rigid tapping.
This value corresponds to Eco level 3 (when bit 0 (ELV) of the parameter No.24303 is set to 0) or Eco level 7 (when bit 0 (ELV) of the parameter No.24303 is set to 1).

When torque override is set to 50%, for example, time constant for acceleration/deceleration gets twice.
Note that 0% is considered as 100%.
If eco-machining is not necessary, please set 0 or 100 in this parameter.

When bit 0 (PWE) of the parameter No.8900 is set to 1, this value can be changed on Eco setting screen.

> **NOTE**
> This parameter requires the option of Energy saving level selecting function.

11399	Conversion factor from power consumption (kWh) to carbon-dioxide emission (kg)

[Input type] Parameter input
[Data type] Word
[Unit of data] 0.001kgCO2/kWh
[Valid data range] 0 to 1000

Specify conversion factor from power consumption (kWh) to carbon-dioxide emission (kg). Adjust the value to correspond to power supplying circumstances in the area where a machine is used.
If 0 or invalid value is specified, 0.555kgCO2/kWh is used as the conversion factor.

> **NOTE**
> This parameter requires the option of Energy saving level selecting function.

	#7	#6	#5	#4	#3	#2	#1	#0
24303							EEP	ELV

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ELV Divisions of Eco levels are
0: 4 levels.
1: 8 levels.

> **NOTE**
> This parameter requires the option of Energy saving level selecting function.
> If this parameter is changed, adjust the parameters Nos. 11397, 11398 and 5260 again.
> If the current Eco level gets invalid because of change in this parameter, the current Eco level is set to Eco level 0.

#1 EEP An Eco level is
0: Not specified in each path.
The Eco level is common for all paths.
1: Specified in each path.

> ⚠ **CAUTION**
> Machine or work can be damaged by out-of-synchronization when Eco levels of spindles in synchronization control mode differ from each other. This is because this function differs the time constant for acceleration/deceleration in spindle synchronization mode.
> Set “0” without fail when using spindle synchronization control between paths.

NOTE
This parameter requires the option of Energy saving level selecting function.
When this parameter is set to "0" from "1", the current Eco level is set to the Eco level of path 1 before changing this parameter.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Power consumption monitoring screen
	Eco setting screen
	Eco monitoring screen
	Power consumption setting screen

13.1.23 Simultaneous Displaying Axis Number Expansion Function

Overview

The number of axis data "current position" and "Distance to go" that are displayed on the same screen can be expanded up to 20.

Explanation

In initial setting, the maximum number of simultaneous displaying axis of current position is for 5. In this function, the maximum number of simultaneous displaying axes can be selected from 5, 10 and 20 axes by setting the parameter No.11305.
The relation of the maximum number of simultaneous displaying axes and the set value of parameter is shown in the following table.

Table. 13.1.23 (a) The related of the maximum number of simultaneous displaying axes and the set value of parameter

Maximum number of simultaneous displaying axes	Parameter (No.11305)
5	0
10	1
20	2

When the number of axes is more than that of the maximum number of simultaneous displaying axis, you can change to the axis coordinates of the remainder by pressing the chapter selection soft key.

(Example)

When 13 axes (X1 to X13) are displayed by each maximum number of simultaneous displaying axes, the display of coordinates is shown in following figures.

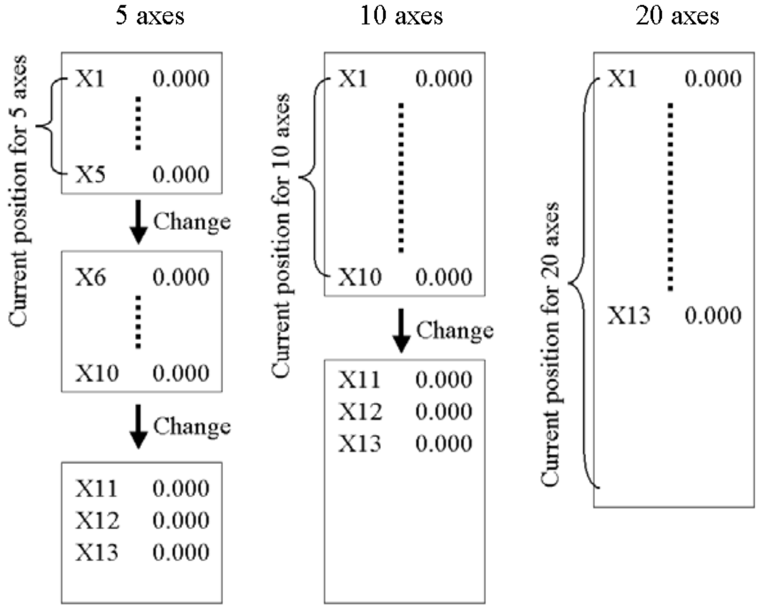


Fig. 13.1.23 (a) Example when current position coordinates for 13 axes are displayed by number of maximum each axes.

When the maximum number of simultaneous displaying axes is 5.

The maximum number of simultaneous displaying axis becomes 5 by setting 0 to parameter No. 11305. When the number of axes exceeds 5, by pressing the chapter selection soft key, axes for the 6th axis and up are displayed.

Fig. 13.1.23 (b) Current position display of 10.4 inch (overall)

Fig. 13.1.23 (c) Current position display of 15 inch (overall)

When the maximum number of simultaneous displaying axes is 10.

The maximum number of simultaneous displaying axis becomes 10 by setting 1 to parameter No. 11305. When the number of axes exceeds 10, by pressing the chapter selection soft key, axes for the 11th axis and up are displayed.

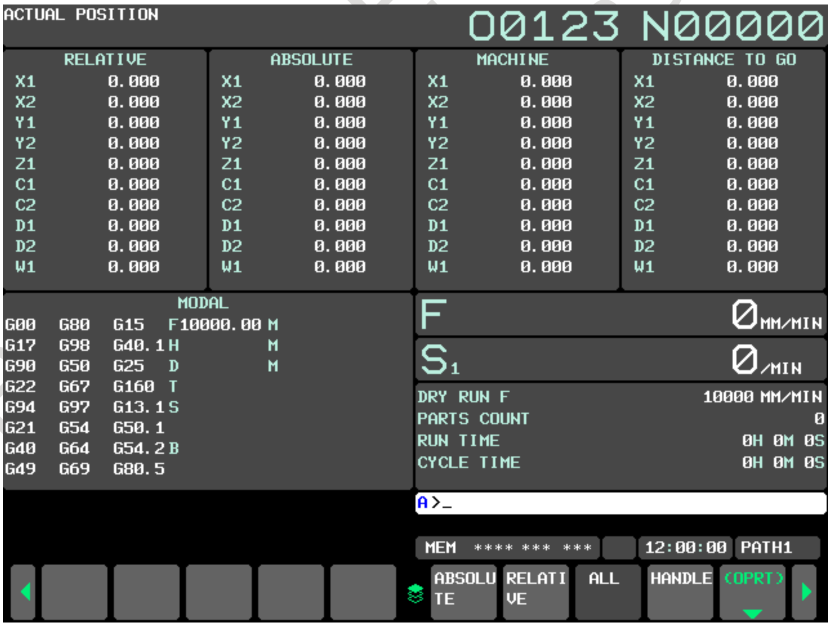


Fig. 13.1.23 (d) Current position display of 10.4 inch (overall)

Fig. 13.1.23 (e) Current position display of 15 inch (overall)

Addition of the chapter selection soft key in 15 inch

The chapter selection soft key of "Absolute" and "Relative" is added setting 1 or 2 to parameter No. 11305.

When you press the chapter selection soft key of "Absolute", absolute coordinate is displayed with the character of 6 times size.

When you press the chapter selection soft key of "Relative", relative coordinate is displayed with the character of 6 times size.

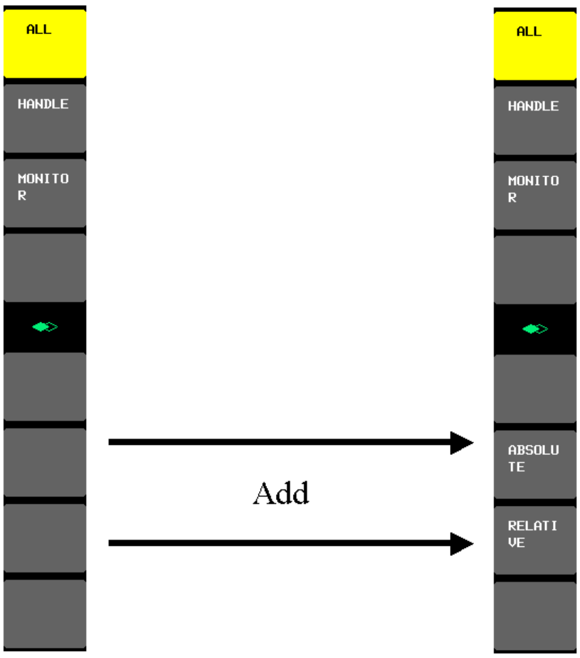


Fig. 13.1.23 (i) Addition of the chapter selection soft key

Fig. 13.1.23 (f) Current position display of 15 inch (relative)

Change of display coordinates

When the maximum number of simultaneous displaying axes is 20 (Parameter No.11305=2), Coordinates displayed in the position display of each screen at the same time are two pairs. The combination of two coordinates is decided by setting the order of displaying four coordinates systems displayed in overall position display screen (parameter No.11307).

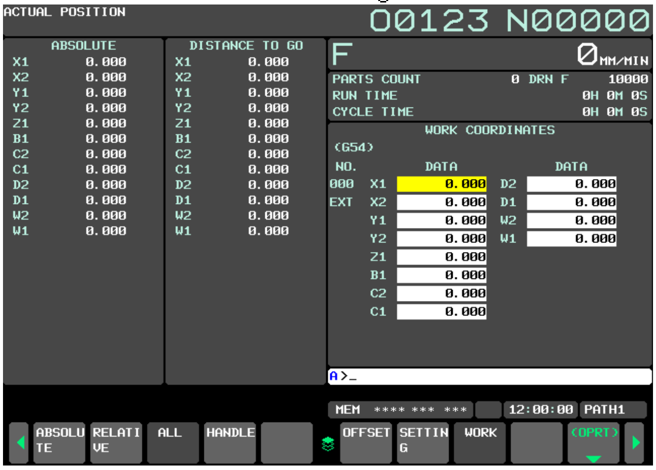
At power-on, the first combination of coordinate is displayed. You can change it

the second one by pressing the chapter selection soft key to selection after SHIFT key. When you do it again, the combination of coordinates is changed to the first one.

(Example)

The workpiece coordinate system setting screen of 10.4 inch. When parameter No.11307 is 4. The maximum number of simultaneous displaying axes is 20.

Absolute coordinates and Distance to go



Relative coordinate and Machine coordinate

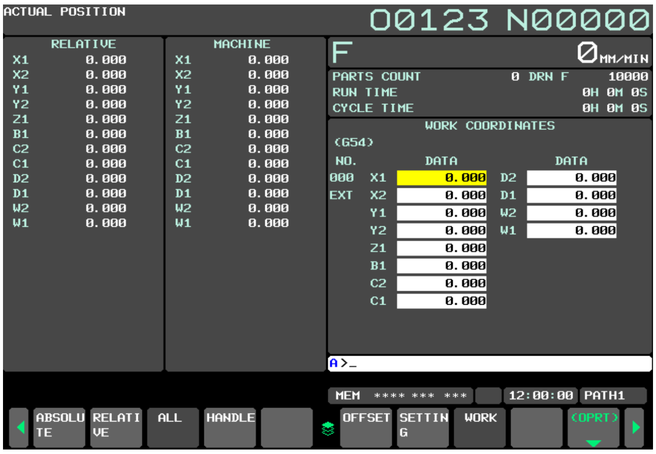


Fig. 13.1.23 (g) Display change example in 10.4 inch.

(Example)

The workpiece coordinate system setting screen of 15 inch. When parameter No.11307 is 4. The maximum number of simultaneous displaying axes is 20.

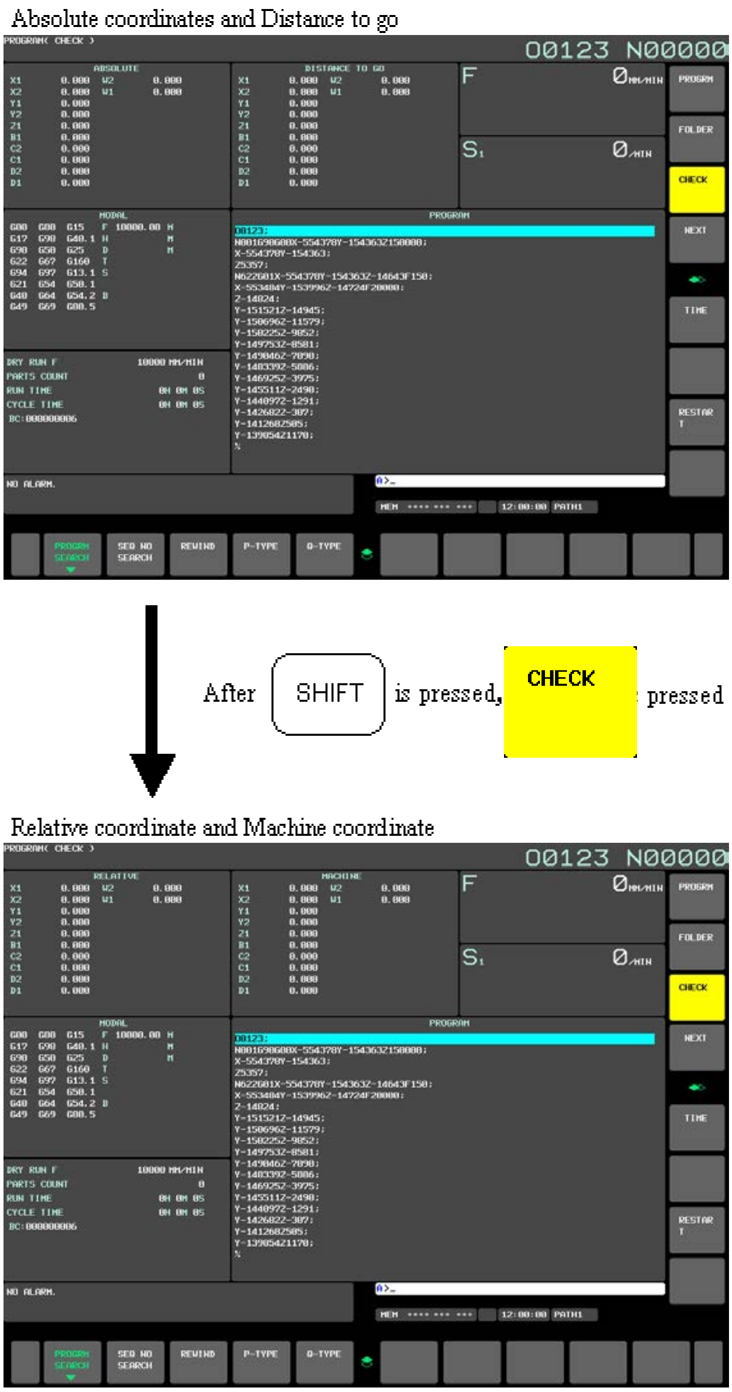


Fig. 13.1.23 (h) Display change example in 15 inch.

About the function for which this function is effective

This function is effective to positional coordinate screen concerning the following functions.

- Handle interruption
- Program restart
- Hypothetical axis
- Operating monitor display

This function is ineffective to positional coordinate screen concerning the following functions

- 3-dimensional manual feed
- Simultaneous display of multiple paths

Parameter

11305	Maximum number of simultaneous displaying axes

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 2

The maximum number of simultaneous displaying axes can be selected from 5, 10 and 20 axes by setting this parameter. The table below indicates the correspondence between the max number of simultaneous displaying axes and the setting value.

Maximum number of simultaneous displaying axes	5	10	20
Setting	0	1	2

When the value other than 1 and 2 is set, the specification of 0 is assumed.

11307	The order of displaying coordinates of current position display

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 5

Set the order of coordinates displayed in the screen of the following size.
10.4inch, 15inch

- Overall Position Display
- Overall Position Display of each screen

When the maximum number of simultaneous displaying axes is 20 (Parameter No.11305=2), Coordinates displayed in the position display of each screen at the same time are two pairs. The combination of two coordinates is decided by setting the order of displaying four coordinates systems displayed in overall position display screen (parameter No.11307).
At power-on, the first combination of coordinate is displayed. You can change it the second one by pressing selection chapter soft key to selection after ⇧ SHIFT key. When you do it again, the combination of coordinates is changed to the first one.
The correspondence of the order of the display and set value of coordinates is as follows.

Setting Value \ Order of displaying coordinates	1	2	3	4
0	Relative coordinates	Absolute coordinates	Machine coordinates	Distance to go
1	Relative coordinates	Machine coordinates	Absolute coordinates	Distance to go
2	Relative coordinates	Distance to go	Absolute coordinates	Machine coordinates
3	Absolute coordinates	Machine coordinates	Relative coordinates	Distance to go
4	Absolute coordinates	Distance to go	Relative coordinates	Machine coordinates
5	Machine coordinates	Distance to go	Relative coordinates	Absolute coordinates

When the value not within the valid data range is set, the specification of 0 is assumed.
When simultaneous display of multiple path is effect (Parameter No.13131≠0, parameter No.13132≧1), this parameter is ineffective.

13.1.24 Warning Function Against Modification of Setting

13.1.24.1 Overview

This function is to protect important setting of parameters, C Language Executor programs, or ladder programs on CNC and to detect unauthorized modifications on them. The following parameters and programs can be protected.

- CNC Parameters (which are selected to be protected)
- C Language Executor program
- Ladder program (each of multi-path PMC programs)
- Dual Check Safety PMC Ladder program

After the registration of above parameters, C Language Executor programs, or ladder programs to this function, those are verified by CNC, when the power of CNC is turned on. If any modification is applied to registered parameters, C Language Executor programs, or ladder programs the signal that means some modification is applied is output. And in this case, it is also possible to generate the alarm.
This is an optional function.

Procedure for protecting parameters, C Language Executor programs, or ladder programs

Procedure

Preparation for registering protected parameters, C Language Executor programs, or ladder programs.

1) Install C Language Executor programs or ladder programs to be protected into the flash ROM.
2) Set the value of parameters to be protected.
3) Edit the ladder program and Dual Check Safety ladder program, if necessary.

Registration of protected parameters, C Language Executor programs, or ladder programs on setting screen.

4) Enter password to enable operations on setting screen. (Refer to 13.1.24.3 and 13.1.24.4)
   By this operation, the following operations 5), 6), 7) are enabled.
5) Change the setting of items, if necessary. (Refer to 13.1.24.2)
6) Select the parameters to be protected. (Refer to 13.1.24.7)
7) Register the parameters, C Language Executor programs, or ladder programs to be protected. (Refer to 13.1.24.3)
   After this operation, if any unauthorized modification to the selected parameters, C Language Executor programs, or ladder programs is detected, CNC outputs signal and alarm.

8) Disable operations on setting screen. (Refer to 13.1.24.3)
Even if the power of CNC is turned off or the screen is switched to other one without the operation 8), operations on the setting screen become disabled automatically

Procedure for cancelling protection of parameters, C Language Executor programs, or ladder programs

Procedure

Cancelling protection of parameters, C Language Executor programs, or ladder programs to be protected on setting screen

1) Enter password to enable operations on setting screen. (Refer to 13.1.24.3 and 13.1.24.4)
By this operation, the following operation 2 is enabled
2) Cancel protection of parameters, C Language Executor programs, or ladder programs. (Refer to 13.1.24.6)
After this operation, the protection is cancelled.
3) Disable operations on the setting screen. (Refer to 13.1.24.3)
Even if the power of CNC is turned off or the screen is switched to other one without this operation, operations on the setting screen become disabled automatically.

13.1.24.2 MODIFICATION WARNING SETTING screen

The operation and displayed information for this function are operated on the MODIFICATION WARNING SETTING screen.

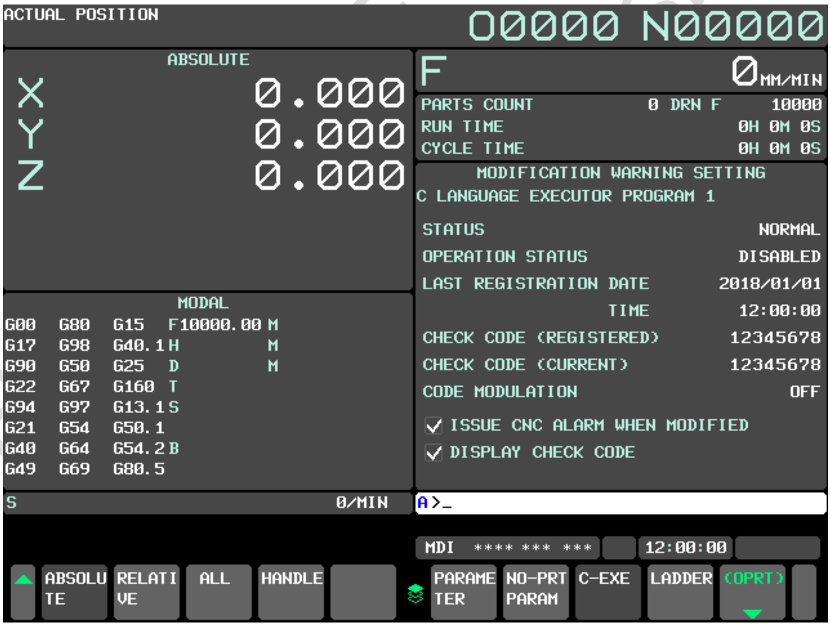


Fig 13.1.24.2 (a) MODIFICATION WARNING SETTING screen

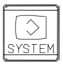
There are separate setting screens for each item to be protected as follows.
You can switch to each setting screen by soft key operation.

MODIFICATION WARNING SETTING screens
- Setting screen for CNC parameters
- Setting screen for non-protected parameters
- Setting screen for C language executor program
- Setting screen for ladder program (Dual Check Safety ladder program is contained.)

> **NOTE**
> 1. You can select any parameters to be protected. Refer to the section "13.1.24.7 Selection of parameters".
> 2. Each ladder program for the 1st – 5th path PMC and Dual check safety PMC can be protected independently. Each ladder program is protected by independent password and settings.

Display setting screen

Procedure

1) Press function key .
2) Press chapter selection soft key [MODIFY WARN].
3) Press the following soft key according to the item which you want to display.

[PARAMETER]	:	CNC parameters
[NO-PRT PARAM]	:	Non-protected parameters
[C-EXE]	:	C Language Executor program
[LADDER]	:	Ladder program (Dual Check Safety ladder program is contained.)

> **NOTE**
> Setting screen for ladder program and setting screen for dual check safety ladder program can be switched with the MDI key [PAGE UP], [PAGE DOWN]. When the key [PAGE UP] / [PAGE DOWN] is pushed, the screen changes in the following sequence.
> 1) 1st path ladder program
> 2) 2nd path ladder program
> 3) 3rd path ladder program
> 4) 4th path ladder program
> 5) 5th path ladder program
> 6) Dual check safety ladder program

Items on setting screen

The following items are displayed on each setting screen.

1) STATUS : Modification status
   NORMAL : Not modified
   MODIFIED : Modified
2) OPERATION STATUS : Status of operations on setting screen
   ENABLED : Operations for changing the items on the setting screen are enabled
   DISABLED : Operations are disabled
3) LAST REGISTRAION DATE and TIME: Registered date
4) CHECK CODE (REGISTERED) : Registered check code
5) CHECK CODE (CURRENT) : Current check code
6) CODE MODULATION : Code-modulation status of C Language Executor program
   ON : Modulated (difficult to analyze)
   OFF : Not modulated (normal C Language Executor program)

> **NOTE**
> 1. Current check code is displayed on "CHECK CODE (CURRENT)"
>   When the parameter is changed, check code after it changes is displayed.
> 2. "CODE MODULATION" is displayed only in the setting screen for C Language Executor program.
>   For details of code modulation, refer to the manual about C Language Executor programming.

Setting items

You can set the following setting items on each setting screen. When you are to change those setting items, enter the password to enable operations on setting screen.

1) When the modification is detected, CNC alarm is "generated" or "not generated".
2) On the setting screen, check code is "displayed" or "not displayed".

Procedure

1) Switch to the setting screen
   Switch to the screen where you want to change the setting by soft key operation.
2) Enable operations on the setting screen
   Press soft key [(OPRT)], then enter password and press the soft key [PASSWORD]. In this case, the entered character is displayed as "*". When the entered password is correct, the OPERATION STATUS becomes "ENABLED".
3) Select the setting item
   Move the cursor to a setting item that you want to change.
4) Change setting
   Change the setting by pressing soft key [ON:1] or [OFF:0].
5) Disable operations on the setting screen
   Press soft key [LOCK] to disable operations on the setting screen.

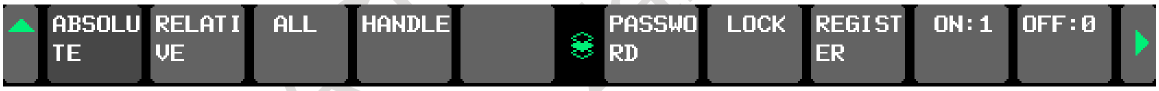


Fig 13.1.24.2 (b) Soft key of MODIFICATION WARNING SETTING screen

> **NOTE**
> 1. The initial password "FFFF" is set to this function when shipping of CNC. Before using this function at the first time, enter the initial password and change password to another one.
> 2. Operation status becomes "DISABLED" form "ENABLED" automatically in the following cases.
>   - When switching to other screens.
>   However, the state of the "operation enabled" is maintained in switching between the screen for CNC PARAMETER SETTING and NON-PROTECTED PARAMETER SETTING screen.
>   - The power of CNC is turned off and on.
> 3. In the state of the "operation disabled", the cursor of the check box and soft key [LOCK], [REGISTER], [ON:1], [OFF:0] are not displayed.

13.1.24.3 Registration and protection parameters, C Language Executor programs, or ladder programs.

You can register parameters, C Language Executor programs, or ladder programs on each setting screen. When a screen is displayed, operations for changing the setting items on the screen are disabled by password. When you change any setting item, you have to enter the password first.
Each setting screen has independent password.

Procedure for registering parameters, C Language Executor programs, or ladder programs

Procedure

1) Switch to the setting screen
   Switch to the screen where you want to change the setting by soft key operation.
2) Enable operations on the setting screen
   Press soft key [(OPRT)], then enter password and press the soft key [PASSWORD]. In this case, the entered character is displayed as “*”. When the entered password is correct, the OPERATION STATUS becomes “ENABLED”.
3) Registration
   Press the soft key [REGISTER].
4) Disable operations on the setting screen
   Press soft key [LOCK] to disable operations on the setting screen.

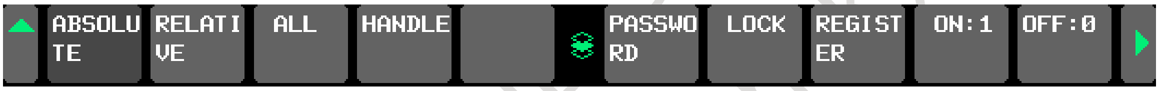


Fig 13.1.24.3 (a) Soft key of MODIFICATION WARNING SETTING screen

> **NOTE**
> 1. The initial password “FFFF” is set to this function when shipping of CNC. Before using this function at the first time, enter the initial password and change password to another one.
> 2. Even after some parameters are registered, you can modify the parameter. But, if you turn off the power of CNC after modifying parameters, CNC will notify the modification of parameters when starting up of CNC. Then, corresponding signal is set and the alarm is generated if you select the setting item to generate the alarm. To remove this notification by signal and alarm, you should restore the registered parameters or register the modified parameters. By this operation, the signal is reset and the alarm is cleared.
> 3. Even if any parameters, C Language Executor programs, or ladder programs are not registered, you can disable operations on setting screen. However in this case, the parameters, C Language Executor programs, or ladder programs are not protected.
> 4. Operation status becomes “DISABLED” form “ENABLED” automatically in the following cases.
>   - When switching to other screens.
>   However, the state of the "operation enabled" is maintained in switching between the screen for CNC PARAMETER SETTING and NON-PROTECTED PARAMETER SETTING screen.
>   - The power of CNC is turned off and on.
> 5. In the state of the "operation disabled", soft key [LOCK], [REGISTER], [ON:1], [OFF:0] are not displayed.

13.1.24.4 Change of password

You can change password on each MODIFICATION WARNING SETTING screen.
When changing the password, you need to enter current password on the setting screen.

> **NOTE**
> Changing password is not necessary to register parameters, C Language Executor programs, or ladder programs. You can repeatedly register parameters, C Language Executor programs, or ladder programs using the same password.

Procedure for changing password

Procedure

1) Switch to the setting screen
   Switch to the screen where you want to change the password by soft key operation.
2) Enable operations on setting screen
   Press soft key [(OPRT)], then enter password and press the soft key [PASSWORD]. In this case, the entered character is displayed as “*”. When the entered password is correct, the OPERATION STATUS becomes “ENABLED”.
3) Enter new password
   Enter new password and press soft key [PASSWORD]. Repeat this operation twice. If these two passwords are same, the password will be changed with the message “PASSWORD CHANGED” on the screen.
   If the two passwords are different, the message “PASSWORD MISMATCH” is displayed, and the password is not changed.
   In either case, the OPERATION STATUS is kept “ENABLED” and you are enabled to perform other operations on the setting screen.
4) Disable operations on setting screen
   Press soft key [LOCK] to disable operations on the setting screen.

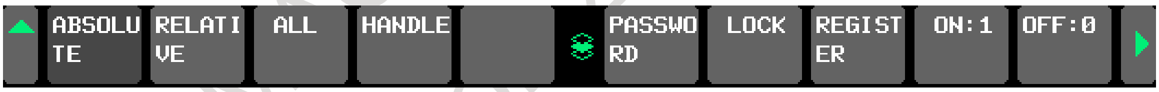


Fig. 13.1.24.4 (a) Soft key of MODIFICATION WARNING SETTING screen

> **NOTE**
> 1. Following characters are available for password.
>   If the lower case alphabetical character, a-z, is set, it is recognized as identical to corresponding upper case alphabetical character, A-Z.
>   - “A” to “Z”, “a” to “z”, “0” to “9”
> 2. The initial password “FFFF” is set to this function when shipping of CNC. When using this function at the first time, enter the initial password to enable operations on setting screen and change password to another one. In other words, when the initial password is set, you cannot execute any operation without changing password.
> 3. You cannot set “FFFF” as a new password.
> 4. In the state of the "operation disabled", soft key [LOCK], [REGISTER], [ON:1], [OFF:0] are not displayed.

13.1.24.5 Notification of the modification of setting

When any parameters, C Language Executor programs, or ladder programs are registered to this function, CNC verifies them at start up of the CNC. When any modification is detected, corresponding signals will be set to notify. In addition, you can enable setting item of this function to generate CNC alarm when any modification is detected.
If you are to remove those notifications of modification, which are resetting signals and clearing CNC alarm, you have to restore the registered parameters, C Language Executor programs, or ladder programs, or you have to register the modified parameters or programs to this function.

> **NOTE**
> When some modification is detected, the notification signals are always set. There is no setting item to specify whether or not to set these signals.

13.1.24.6 Unregistration and canceling protection of parameters, C Language Executor programs, or ladder programs

If you would like to cancel protection of parameters, C Language Executor programs, or ladder programs, unregister the check codes of them on each setting screen. When a screen is displayed, operations for changing the setting items on the screen are disabled by password. When you change any setting item, you have to enter the password first.
Each setting screen has independent password.

Procedure for unregistering of parameters, C Language Executor programs, or ladder programs

Procedure

1) Switch to the setting screen.
   Switch to the screen where you want to change the setting by soft key operation.
2) Enable operations on the setting screen.
   Software key [(OPRT)] is pushed.
   Enter password and press the soft key [PASSWORD]. In this case, the entered character is displayed as “*”.
   When the entered password is correct, the operation status becomes “ENABLED”.
3) Unregistration.
   Press the continuous menu key and press the soft key [UNREGISTER].
4) Disable operations on the setting screen.
   Press soft key [LOCK] to disable operations on the setting screen.

Fig. 13.1.24.6 (a) Soft key of MODIFICATION WARNING SETTING for parameters (Page 1, 2)

NOTE

1. When unregistering, the password is maintained.
2. Operation status becomes “DISABLED” form “ENABLED” automatically in the following cases.
   - When switching to other screens.
     However, the state of the "operation enabled" is maintained in switching between the screen for CNC PARAMETER SETTING and NON-PROTECTED PARAMETER SETTING screen.
   - The power of CNC is turned off and on.
3. In the state of the "operation disabled", soft key [LOCK], [REGISTER], [ON:1], [OFF:0] are not displayed.

13.1.24.7 Selection of parameters

Any parameters can be selected to be protected or not to be protected by definition of non-protected parameters from all of the parameters. Maximum 50 sets of the range of non-protected parameters can be defined.

Parameters not to be protected can be defined on NON-PROTECTED PARAMETER SETTING screen.

Before the setting on this screen, the operations have to be enabled by entering the password on MODIFICATION WARNING SETTING screen for parameters. If the operations on the setting screen are not enabled, you can not define the non-protected parameters.

Moreover, the parameter changed by the system is excluded from the protected parameter.

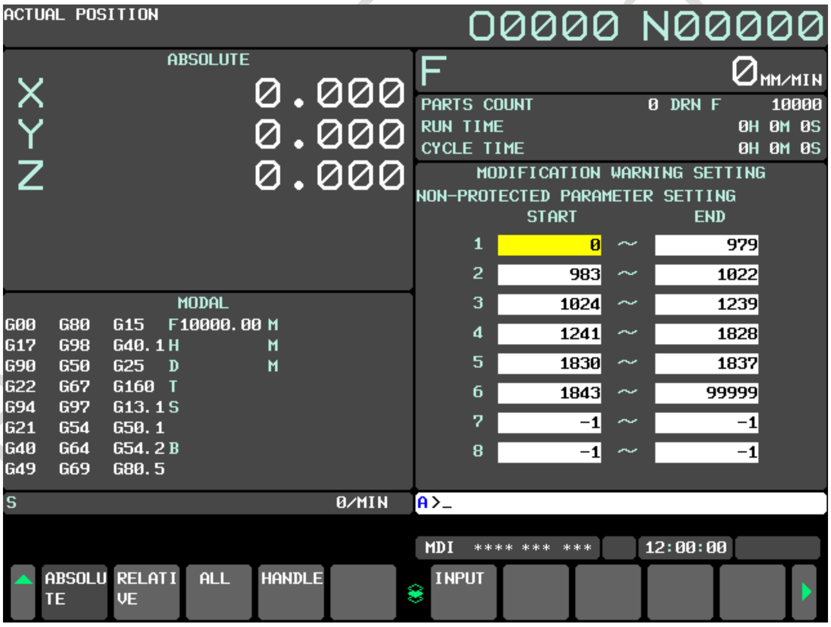


Fig 13.1.24.7 (a) NON-PROTECTED PARAMETER SETTING screen

Setting non-protected parameters

Procedure

1) Change to MDI mode.
2) Enable operations on the setting screen for parameters.
3) Press the return menu key.
4) Press the soft key [NO-PRT PARAM]. NON-PROTECTED PARAMETER SETTING screen is displayed.
5) Move cursor by cursor key ↑ ↓ ← → and page key PAGE ↑ PAGE ↓, then set the range of non-protected parameters to “START NO.” and “END NO” by soft key [INPUT] or [+INPUT].

6) If all of the setting has been finished, press soft key [RETURN] to switch to MODIFICATION WARNING SETTING screen for parameters.
7) Press soft key [REGISTER] to calculate check code from current parameter values. Then the calculated check code is displayed on the item “CHECK CODE(REGISTERED)” and “CHECK CODE(CURRENT)”
8) Press soft key [LOCK] to disable operations on setting screen.

Fig 13.1.24.7 (b) Soft key of MODIFICATION WARNING SETTING screen for parameters (Page 1,2)

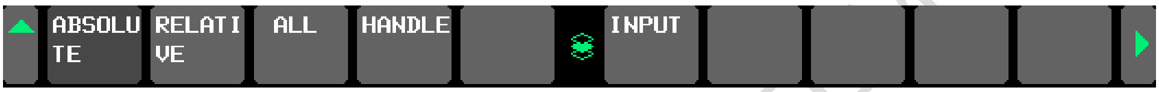


Fig 13.1.24.7 (c) Soft key of NON-PROTECTED PARAMETER SETTING screen (Page 1)

> **NOTE**
> 1 Even if operations on the setting screen for parameters are prohibited, the NON-PROTECTED PARAMETER SETTING screen can be displayed. But, non-protected parameters cannot be set. (Cursor and soft key [INPUT] are not displayed.)
> 2. Operation status becomes “ENABLED” form “DISABLED” automatically in the following cases.
> - When switching to other screens.
>   However, the state of the "operation enabled" is maintained in switching between the screen for CNC PARAMETER SETTING and NON-PROTECTED PARAMETER SETTING screen.
> - The power of CNC is turned off and on.

Setting file of non-protected parameters

Non-protected parameters can be defined also by input of a text file describing the ranges of non-protected parameters in designated format from external I/O devices.
Before input of the file from external I/O device, enable the operations on the screen by entering the correct password on MODIFICATION WARNING SETTING screen for parameters.
And setting of non-protected parameters can be output to external I/O devices. In case of output, you need not to enter the password.
The default name of setting file of non-protected parameters is “MDPRMSET.TXT”.

Example
In case of protecting the following parameters
No.980 - No.982
No.1023
No.1240
N0.1829
No.1838 - No.1842

```
%
G10L99;
N1S0E979;
N2S983E1022;
N3S1024E1239;
N4S1241E1828;
N5S1830E1837;
N6S1843E99999;
G11;
%
```

Input the setting file of non-protected parameters

Procedure

1) Make sure the external I/O device is ready for input.
2) Change to EDIT mode by operating the machine operator's panel or enter emergency stop state.
3) Enable operations on the setting screen for parameters.
4) Press the return menu key.
5) Press the soft key [NO-PRT PARAM]. NON-PROTECTED PARAMETER SETTING screen is displayed.
6) Press the continuous menu key , and then press soft key [F INPUT].
7) Type the name of the file that you want to input.
   If the input file name is omitted, default input file name "MDPRMSET.TXT" is assumed.
8) Press the soft key [EXEC].
   This starts input, and "INPUT" blinks in the lower right part of the screen. When the input operation ends, the "INPUT" indication disappears.
9) After input has been finished, press the return menu key.
10) Press the soft key [PARAMETER]. The setting screen for parameters is displayed.
11) Press the soft key [REGISTER] to calculate the check code.
12) Press the soft key [LOCK] to disable operations.

Fig 13.1.24.7 (d) Soft key of NON-PROTECTED PARAMETER SETTING screen (Page2)

NOTE
In the state of the "operation disabled", soft key [F input] is not displayed.

Output the setting file of non-protected parameters

Procedure

1) Make sure the external I/O device is ready for output.
2) Change to EDIT mode by operating the machine operator's panel or enter emergency stop state.
3) Press the soft key [NO-PRT PARAM]. NON-PROTECTED PARAMETER SETTING screen is displayed.
4) Press the continuous menu key , and then press soft key [F OUTPUT].
5) Type the name of the file that you want to output.
   If the output file name is omitted, default output file name "MDPRMSET.TXT" is assumed.
6) Press the soft key [EXEC].
   This starts input, and "OUTPUT" blinks in the lower right part of the screen. When the input operation ends, the "OUTPUT" indication disappears.
7) After output has been finished, press the return menu key.

Fig13.1.24.7 (e) Soft key of NON-PROTECTED PARAMETER SETTING screen (Page2)

Parameters excluded from protected parameters

Table 13.1.24.7 (a) Parameters excluded from protected parameters

Parameter No.	Description
1244	Coordinates of the floating reference position for each axis
1301	Stored stroke check
1320 to 1321	Coordinates at the boundary of stored stroke check 1
1330 to 1348	Chuck and tail stock barrier
3233	Program edit
3281	Displaying language
3290	Offset
4803, 4840 to 4843	Spindle position
4911 to 4914	Spindle speed fluctuation detection
5130	Cutting value (chamfering value) in thread cutting cycles G92 and G76
5132 to 5133	Depth of cut and escaping amount of multiple repetitive canned cycles G71 and G72
5134	Clearance value of multiple repetitive canned cycle G71 and G72
5135	Escaping amount of X axis in multiple repetitive canned cycle G73
5136	Escaping amount of Z axis in multiple repetitive canned cycle G73
5137	Divide number of multiple repetitive canned cycle G73
5139	Return amount in multiple repetitive canned cycle G74 and G75
5140	Minimum depth of cut in multiple repetitive turning canned cycle G76
5141	Finishing allowance in multiple repetitive turning canned cycle G76
5142	Repetition count of final finishing in multiple repetitive turning canned cycle G76
5143	Tool nose angle in multiple repetitive turning canned cycle G76
5402	Manual handle interruption
5660 to 5677	Flexible synchronization control
5680 to 5687, 5690 to 5693	Flexible synchronization control
6404	Manual handle
6510	Specifying the graphic coordinate system
6581 to 6595	Color number of character
6750	Integrated value of power-on period
7220 to 7283	Name of general-purpose switch on software operator's panel
7310	Program restart
8210	Slant angle
8370 to 8374	Oscillation
8860 to 8861	Trouble diagnosis function
8900	PWE
8940 to 8949	Initial screen title character code
10000 to 10329	Tool offsets
10331 to 10332	External workpiece zero point offsets
10421 to 10435, 10461 to 10475	Screen display colors
10601 to 10719	Waveform diagnosis
11221	Tilted working plane indexing
11309	Menu number selected in pattern menu display
11318	Program list screen
11329 to 11331, 11334 to 11337, 11341 to 11349	Dynamic graphic display function
11355 to 11356	Display
11358 to 11359	Parameter check sum
11364	Program edit
11369	Warning function against modification of setting

Parameter No.	Description
11496	Measurement result of thread groove
11501	Machine lock shift value check
11505	Code of Input/Output of data of USB memory
12801 to 12820, 12841 to 12860	Operation history signal selection
13151	Serial number for the files output to a memory card by external output command DPRNT/BPRNT
13203 to 13204, 13206	Tool management function
13420, 13425 to 13440	Flexible synchronization control
14500 to 14637	SERVO GUIDE Mate
14717	C axis’s number in simulation (for MGi)

Table 13.1.24.7 (b) Parameters excluded from protected parameters (Machining condition selecting function is valid)

Parameter No.	Description
1432	Maximum cutting feedrate for all axes in the look-ahead acceleration / deceleration before interpolation
1660	Maximum allowable acceleration rate in look-ahead acceleration/deceleration before interpolation for each axis
1735	Maximum allowable acceleration rate for the deceleration function based on acceleration in circular interpolation for each axis (Only at the setting of No.13600#0=0)
1737	Maximum allowable acceleration rate for the deceleration function based on acceleration in AI contour control for each axis
1769	Time constant for acceleration/deceleration after cutting feed interpolation in the look-ahead acceleration/deceleration before interpolation mode
1772	Acceleration change time of look-ahead bell-shaped acceleration/deceleration before interpolation
1783	Maximum allowable feedrate difference for feedrate determination based on corner feedrate difference
1788	Maximum allowable acceleration change rate in feedrate determination based on acceleration change for each axis
1789	Maximum allowable acceleration change rate in feedrate determination based on acceleration change for each axis (linear interpolation)
1790	Ratio of change time of the rate of change of acceleration in look-ahead smooth bell-shaped acceleration/deceleration before interpolation
13634	Current precision level in AI contour control

Table 13.1.24.7 (c) Parameters for setting the parameter numbers of parameters to be excluded from protected parameters (Machining condition selecting function is valid)

Parameter explanation
Parameter whose number is set in parameter No.13628 (Parameter number corresponding to arbitrary item 1 when AI contour control is used)
Parameter whose number is set in parameter No.13629 (Parameter number corresponding to arbitrary item 2 when AI contour control is used)

NOTE

1. If machining condition selecting function is invalid, parameters in table 13.1.24.7 (b) and table 13.1.24.7 (c) are contained in protected parameters
2. If machining condition selecting function is invalid, parameters in table 13.1.24.7 (d) are contained in protected parameters.
3. The parameters of which numbers are set in parameter Nos. 13628 and 13629 are excluded, but parameter Nos. 13628 and 13629 are not excluded.
4. Setting parameters and locked parameters are contained in non-protected parameters.
5. When the CNC software edition is changed, the alarm (DS5350) might be generated. In this case, register the parameter again.

13.1.24.8 Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
11369						MDP	MDL	MDC

[Data type] Bit

#0 MDC MODIFICATION WARNING SETTING screen for C Language Executor is
0: not displayed.
1: displayed.

#1 MDL MODIFICATION WARNING SETTING screen for PMC ladder is
0: not displayed.
1: displayed.

#2 MDP MODIFICATION WARNING SETTING screen for parameters and non-protected parameters is
0: not displayed.
1: displayed.

13.1.24.9 Signals

Notification signal for modification of C Language Executor program
CDCEX <F0558.0>

[Classification] Output signal
[Function] Notify the modification of C Language Executor program
[Output condition] When this signal becomes 0, the program is not modified or not registered
When this signal becomes 1, the registered program is modified

Notification signal for modification of 1st path PMC Ladder program
CDLAD1 <F0558.1>

[Classification] Output signal
[Function] Notify the modification of 1st path PMC Ladder program
[Output condition] When this signal becomes 0, the program is not modified or not registered
When this signal becomes 1, the registered program is modified

Notification signal for modification of 2nd path PMC Ladder program
CDLAD2 <F0558.2>

[Classification] Output signal
[Function] Notify the modification of 2nd path PMC Ladder program

[Output condition] When this signal becomes 0, the program is not modified or not registered
When this signal becomes 1, the registered program is modified

---

Notification signal for modification of 3rd path PMC Ladder program
CDLAD3 <F0558.3>

[Classification] Output signal
[Function] Notify the modification of 3rd path PMC Ladder program
[Output condition] When this signal becomes 0, the program is not modified or not registered
When this signal becomes 1, the registered program is modified

---

Notification signal for modification of Dual Check Safety PMC Ladder program
CDDCL <F0558.4>

[Classification] Output signal
[Function] Notify the modification of Dual Check Safety PMC Ladder program
[Output condition] When this signal becomes 0, the program is not modified or not registered
When this signal becomes 1, the registered program is modified

---

Notification signal for modification of CNC parameter
CDPRM <F0558.5>

[Classification] Output signal
[Function] Notify the modification of selected CNC parameters
[Output condition] When this signal becomes 0, any selected CNC parameters are not modified or not registered
When this signal becomes 1, some selected CNC parameters are modified

---

Notification signal for modification of 4th path PMC Ladder program
CDLAD4 <F0558.6>

[Classification] Output signal
[Function] Notify the modification of 4th path PMC Ladder program
[Output condition] When this signal becomes 0, the program is not modified or not registered
When this signal becomes 1, the registered program is modified

---

Notification signal for modification of 5th path PMC Ladder program
CDLAD5 <F0558.7>

[Classification] Output signal
[Function] Notify the modification of 5th path PMC Ladder program
[Output condition] When this signal becomes 0, the program is not modified or not registered
When this signal becomes 1, the registered program is modified

---

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F0558	CDLAD5	CDLAD4	CDPRM	CDDCL	CDLAD3	CDLAD2	CDLAD1	CDCEX

13.1.24.10 Alarm and Message

Number	Message	Contents
DS5345	C-EXECUTOR VERIFY ERROR	C Language Executor program has been modified. Restore the registered program or register the modified program.
DS5346 DS5347 DS5348 DS5352 DS5353	LADDER PROGRAM VERIFY ERROR	Ladder program has been modified. Restore the registered ladder program or register the modified ladder program.
DS5349	DCS LADDER PROGRAM VERIFY ERROR	Ladder program of Dual Check Safety PMC has been modified. Restore the registered ladder program or register the modified ladder program.
DS5350	PARAMETER VERIFY ERROR	CNC parameter has been modified. Restore the registered parameters or register the modified parameters.

NOTE

The following DS alarm numbers are corresponding to ladder program of multi-path PMC

DS5346 = 1st path PMC ladder program
DS5347 = 2nd path PMC ladder program
DS5348 = 3rd path PMC ladder program
DS5352 = 4th path PMC ladder program
DS5353 = 5th path PMC ladder program

13.1.25 Title Display Function in Initial Screen

Overview

The character string (up to 10 characters) set by the parameter is displayed on the initial screen immediately after power-on.

Details

The character string set by parameter No.8940 to No.8949 can be displayed on the initial screen immediately after power-on as shown in the following figure 13.1.25 (a). ("ABCDE01234" is displayed in the example of figure)

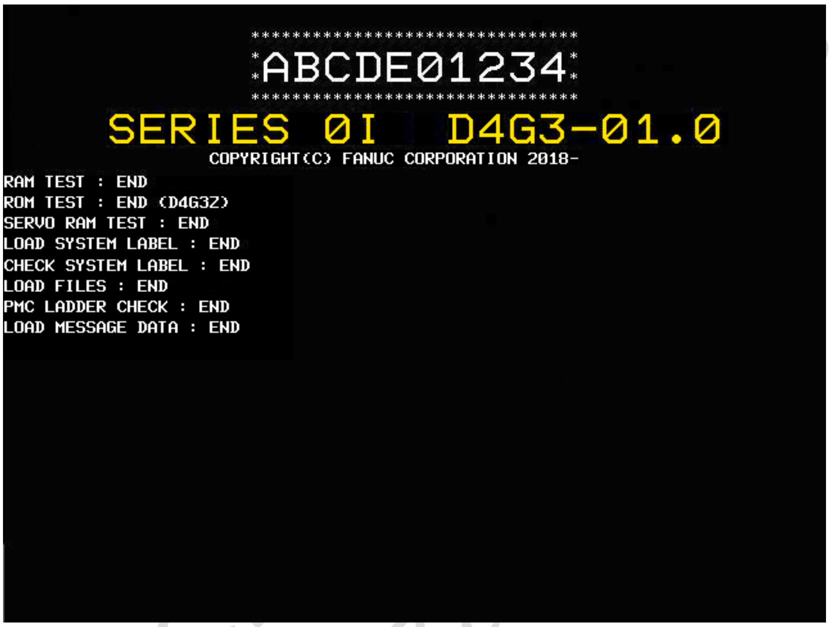


Fig.13.1.25 (a) Title display in initial screen（10.4” display unit）

The character string is enclosed with the frame (* mark) and displayed. The color of the character is only white. It is possible to display up to ten characters, and the character that can be used is the following characters.

0 to 9
A to Z
“-”（Negative sign）
“.”（period）
“ ”（space）

The frame becomes small in proportion to the number of characters when character strings are less than 10 characters.
Character string, series name, and COPYRIGHT are displayed at the center for the 10.4” display unit. However, in 15” display unit, it is displayed as shown in the following figure 13.1.25 (b).

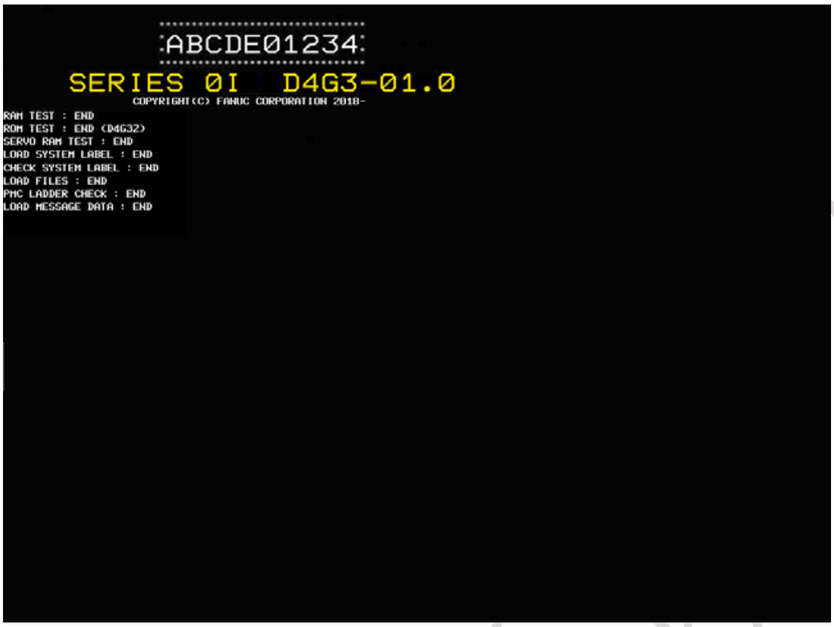


Fig.13.1.25 (b) Title display in initial screen（15” display unit）

When parameter No.8940-No.8949 is not set (When the parameter is all 0), the character string is not displayed. A conventional initial screen is displayed as shown in Figure 13.1.25 (c).

Fig.13.1.25 (c) Initial screen（conventional display）

NOTE

1. The character string is displayed also on the IPL screen. However, when the IPL menu screen ends, a conventional screen is displayed.
2. When the parameter is set by using "SRAM RESTORE" of the boot menu, the set character string is not displayed on an initial screen immediately after restore. It is necessary to restart CNC to display the character string.

Parameters

8940	**Initial screen title character code 1**
8941	**Initial screen title character code 2**
to	**to**
8949	**Initial screen title character code 10**

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0,32,45,46,48 to 57,65 to 90

This parameter sets the character codes of the character string to be displayed on the initial screen immediately after power-on. If the number of characters to be displayed is less than 10, the parameters corresponding to the space exceeding the character length are set to 0.

The set character code is as follows.

A : 65	N : 78	0 : 48	
B : 66	O : 79	1 : 49	
C : 67	P : 80	2 : 50	
D : 68	Q : 81	3 : 51	
E : 69	R : 82	4 : 52	
F : 70	S : 83	5 : 53	
G : 71	T : 84	6 : 54	
H : 72	U : 85	7 : 55	
I : 73	V : 86	8 : 56	
J : 74	W : 87	9 : 57	
K : 75	X : 88	: 32	(Space)
L : 76	Y : 89	- : 45	(Negative sign)
M : 77	Z : 90	. : 46	(Period)
		: 0	(Null character)

When the codes other than the character that can be used are set, space is displayed.
When 10 parameters are all 0, this function is not executed.

Use example

For example, the parameter is set as follows when displaying as "FANUC_NC". ("_" is space)

Parameter No.	Input value
8940	70
8941	65
8942	78
8943	85
8944	67
8945	0
8946	78
8947	67
8948	0
8949	0

Character-code 0 is basically considered to be space, and it is treated as a null character when there is only character-code 0 since 0.
The input value “0” of parameter No.8945 is treated as space because there are character-codes other than 0 since No.8946.
The input value “0” of parameter No.8948 is treated as a null character because there is only character-code 0 since No.8949.

13.1.26 One-touch menu

Overview

By using One-touch menu, you can allocate shortcut key of screen witch you use frequently.
This function can be used with 10.4”, 15” and 19” display.

Signal

One-touch menu display signal WMDS <F0528.7>

[Classification] Output signal
[Function] This signal indicates that one-touch menu is being displayed.
[Output cond.] The signal is set to “1” when:
- One-touch menu is displayed.

The signal is set to “0” when:
- One-touch menu is not displayed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F0528	WMDS							

NOTE
This signal is set to “1” only while the one-touch menu is displayed.
If machine operation menu is displayed, this signal always becomes “0”.
Following display unit that can display one-touch menu is necessary for this signal.
10.4-inch, 15-inch, 19-inch

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3208			PSC	OFY	NOS			SKY

[Input type] Setting input
[Data type] Bit

#3 NOS The one-touch menu is
0: Displayed.
1: Not displayed.

3321	Screen number assigned to the 1st vertical soft key
to	to
3336	Screen number assigned to the 16th vertical soft key

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 10000
Assign a screen number to be displayed as a shortcut to a vertical soft key.

The 1st to 8th vertical soft keys are displayed on page 1, and the 9th to 16th vertical soft keys are displayed on page 2.
When specifying page 2, be sure to specify "Display of next page" on each page.
When not specifying page 2, set 0 for the 9th to 16th soft keys.

In this case, page 2 is not used, so that "Display of next page" need not be specified on page 1.

If all of these parameters are 0 when turning on the power supply, one-touch menu of the default table is displayed.

NOTE
1 When screen numbers are specified in at least one of CNC parameters Nos. 3321 to 3336, the setting is valid. In the case, nothing is displayed in the vertical soft-key where 0 is specified in the parameter.
2 In order to display the default one-touch menu again, it is needed to turn off power after all of these parameters are set to 0.

24300	Time for One-touch menu to be closed (before switching screen)

[Input type] Parameter input
[Data type] Byte
[Unit of data] sec
[Valid data range] 0 to 127

The One-touch menu is automatically closed when the One-touch menu is not operated during the time set by this parameter before switching the screen with the One-touch menu.

24301	Time for One-touch menu to be closed (after switching screen)

[Input type] Parameter input
[Data type] Byte
[Unit of data] sec
[Valid data range] -1 to 127

The One-touch menu is automatically closed when the One-touch menu is not operated during the time set by this parameter after switching the screen with the One-touch menu.
If -1 is set, the one-touch menu closes at once after switching the screen.

Ex1)
In case that until switching the screen from One-touch menu is open, automatic close is invalid.
One-touch menu is closed automatically 5 second later after switching the screen.
No.24300 = 0, No.24301 = 5

Ex2)
In case that when One-touch menu is closed automatically with no operation of One-touch menu 10 second from open.
One-touch menu is closed automatically just after switching the screen.
No.24300 = 10, No.24301 = -1

NOTE
1 If 0 is set to parameter No.24300 and No.24301, One-touch menu is not closed automatically.
2 When the One-touch menu switched to the screen being displayed now is pressed, parameter No.24301 is applied though the screen doesn't switch.

13.2 EDIT

13.2.1 Memory Protection Keys

Overview

Memory protection keys can be provided so as not to inadvertently store, change, or delete programs, offset values, parameters, settings, and so forth.
For a multi-path system, protection keys are common to all paths.

Signal

Memory protection signals KEY1 to KEY4<G0046.3 to G0046.6>

[Classification] Input signal

[Function] These signals allow MDI panel operations that change the memory contents. There are the following four signals. The target memory contents depend on the setting of bit 7 (KEY) of parameter No.3290.

When the KEY is "0" :

KEY1 : Allows the input of tool offset values, workpiece origin offsets, and workpiece coordinate system shift values.
KEY2 : Allows the setting of data input, macro variable input, and tool life management data input.
KEY3 : Allows program registration and editing.
KEY4 : Allows PMC data (counter data table).

When the KEY is "1" :

KEY1 : Allows program registration, program editing, PMC data input.
KEY2 to KEY4 : Not used.

[Operation] If a memory protection signal is set to "0", the corresponding operation is disabled.
If a memory protection signal is "1", the corresponding operation is enabled.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0046		KEY4	KEY3	KEY2	KEY1			

NOTE
For a multi-path system, signal addresses are common to all paths.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3290	KEY							

[Input type] Parameter input
[Data type] Bit path

#7 KEY For memory protection keys:
0: The KEY1, KEY2, KEY3, and KEY4 signals are used.
1: Only the KEY1 signal is used.

NOTE

1 The functions of the signals depend on whether KEY=”0” or KEY=”1”.
When KEY = “0”:
- KEY1: Enables a tool offset value, workpiece zero point offset value, and workpiece shift value to be input.
- KEY2: Enables setting data, macro variables, and tool life management value to be input.
- KEY3: Enables program registration and editing.
- KEY4: Enables PMC data (counter and data table) to be input.

When KEY = “1”:
- KEY1: Enables program registration and editing, and enables PMC parameter input.
- KEY2 to KEY4: Not used

2 When a multi-path system is used, the setting for path 1 is followed.

13.2.2 Memory Protection Signal for CNC Parameter

Overview

It is possible to use a signal to enable and disable parameter writing. This function is enabled and disabled by setting bit 0 (PKY) of parameter No. 3299, and the signal KEYP<G0046.0> is used to enable and disable parameter input.
Conventionally, parameter writing was enabled and disabled on the setting screen.

Signal

Memory protection signal KEYP<G0046.0>

[Classification] Input signal
[Function] Enables or disables operation on CNC parameters from the MDI panel.
[Operation] If this signal is set to “0”, input of CNC parameters is disabled.
If this signal is set to “1”, input of CNC parameters is enabled.
This signal is valid only when bit 0 (PKY) of parameter No. 3299 is set to 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0046								KEYP

NOTE
For a multi-path system, the signal address is common to all paths.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3299								PKY

[Input type] Setting input
[Data type] Bit

#0 PKY "PARAMETER WRITE" is:
0: Set on the setting screen (bit 0 (PWE) of setting parameter No. 8900).
1: Set by the memory protection signal KEYP<G0046.0>.

Notes

- When bit 0 (PKY) of parameter No. 3299 is set to 1, “PARAMETER WRITE” on the setting screen cannot be set.
- When bit 0 (PKY) of parameter No. 3299 has been set to 1 and bit 0 (PWE) of parameter No. 8900 has been set to 0 at power-on, setting the signal KEYP<G0046.0> to “1” in the operation mode causes alarm SW0100 and stops operation.

13.2.3 MDI Key Setting

Overview

Because the type of the MDI keys of the CNC is determined automatically, no additional setting is required for the MDI keys. When the machine tool builder's own MDI keys are connected, valid key input is made possible by parameter setting.

Parameter

3160	Setting of MDI unit type

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 4

Set the type of an MDI unit when the type of an MDI unit is not automatically identified.

Setting value	Type
0	Depends on the system type and display unit type.
1	Standard MDI unit for the lathe system
2	Standard MDI unit for the machining center system

When 0 is set in this parameter, the type of a MDI unit is determined as follows:

Type of path control	Type of indicator	Type
When the type for the lathe system is used with path 1	Type of 12 horizontal soft keys	Standard MDI unit for the lathe system
When the type for the machining center system is used with path 1	Type of 12 horizontal soft keys	Standard MDI unit for the machining center system

13.3 MULTI PATH DISPLAY AND EDIT

13.3.1 Multi Path Display

Overview

In a multi path system, operations such as program creation and data input are performed with a display/operation target path selected by switching. For path selection, the path select signals are used. For a selected path, MDI key operation is performed.
By parameter setting, a selection can be made to display only one path on the screen at a time or display multiple paths on the screen simultaneously. When multiple paths are displayed simultaneously, the order of path display can be specified according to the machine configuration and layout.
Moreover, a selection can be made by a parameter to display the same screen at path switching or to store the last displayed screen of each path and display the last displayed screen of a selected path.

Explanation

- Multi path simultaneous display

Information about multiple paths can be displayed on one screen simultaneously.
Multi path information can be displayed simultaneously on the following screens:

- Absolute position screen
- Relative position screen
- Overall position display
- Program screen
- Program check screen
- Tool path graphic screen

MDI key operation is performed for a screen selected with the path select signals.
The maximum number of paths that can be displayed simultaneously depends on the size of the display unit as indicated in the Table 13.3.1 (a).

Table 13.3.1 (a)

Size of display unit	Maximum number of simultaneously displayable paths
7.2, 8.4, or 10.4”	3 paths
15”	4 paths

Setting of multi path simultaneous display

- Simultaneous display group

When paths more than the maximum number of simultaneously displayable paths are involved, the paths can be divided into several path groups for display. Those groups are referred to as multi path simultaneous display groups.

Example)

When four paths are handled on a 10.4” display unit, all paths cannot be displayed simultaneously. So, the paths are divided into two simultaneous display groups: a group consisting of paths 1, 2, and 3, and a group consisting of paths 4.

Table 13.3.1 (b)

Path number	Simultaneous display group number
1, 2, 3	1
4	2

Use parameter No. 13131 to set a multi path simultaneous group number. Parameter No. 13131 is a path-type parameter. In this parameter, specify a successive number not less than 1.
In this example, specify 1 in parameter No. 13131 for paths 1, 2, and 3, and specify 2 in parameter No. 13131 for paths 4.

- Display order number

In a multi path simultaneous display group, the order of path display can be specified.
Path information can be displayed in a desired order by specifying display order numbers.

Example)

When six paths are involved and divided into two groups and group 1 is displayed in the order from path 3 to path 2 to path 1 and group 2 is displayed in the order from path 4, specify display order numbers in each multi path simultaneous display group.

Table 13.3.1 (c)

Number of CNC paths	Path	Display group number (Parameter No. 13131)	Display order number (Parameter No. 13132)	Screen display (The numbers in the illustration below represent path numbers.)
4 paths	Path 1	1	3	3 | 2 | 1
	Path 2	1	2	
	Path 3	1	1	
	Path 4	2	1	4
	Path 5			
	Path 6			

Use parameter No. 13132 to set a display order number. Parameter No. 13132 is a path-type parameter. In this parameter, specify a successive number not less than 1 for each of the paths belonging to a multi path simultaneous display group.
In this example, specify 1, 2, and 3 for paths 3, 2, and 1 in display group number 1 respectively, and specify 1 for paths 4 in display group number 2 respectively.

- Example of setting simultaneous display groups and display order numbers

Number of CNC paths	Path	Display group number (Parameter No. 13131)	Display order number (Parameter No. 13132)	Screen display (The numbers in the illustration below represent path numbers.)	
1 path (*)	Path 1	0	0	Group 1	1
3 paths	Path 1	1	1	Group 1	1 | 2 | 3
	Path 2	1	2		
	Path 3	1	3		
	Path 1	1	1	Group 1	1
	Path 2	2	1	Group 2	2
	Path 3	3	1	Group 3	3
	Path 1	1	2	Group 1	2 | 1
	Path 2	1	1		
	Path 3	2	1	Group 2	3
4 paths	Path 1	1	2	Group 1	2 | 1 | 4
	Path 2	1	1		
	Path 3	2	1	Group 2	3
	Path 4	1	3		

NOTE
When the number of CNC paths is 1, multi path simultaneous display is disabled. In this case, parameters Nos. 13131 and 13132 need not be set.

Switching of simultaneous display group

When multi path simultaneous display groups are set, the screen display can be switched from the currently displayed path group to another by performing a group switching operation.
Two methods described below can be used to switch from one group to another in multi path simultaneous display.

- Switching by path select signal

By setting bit 0 (PGR) of parameter No. 11304 to 1, the display group switching function based on the path select signals can be enabled. The simultaneous display group including a path selected with the path select signals is displayed.

Number of CNC paths	Path	Display group number (Parameter No. 13131)	Display order number (Parameter No. 13132)	Path selected with the path select signals	Screen display (The numbers in the illustration below represent path numbers.) (Box in heavy lines: Selected path)
3 paths	Path 1	1	2	1	$\boxed{1}$ 2
	Path 2	1	1	2	1 $\boxed{2}$
	Path 3	2	1	3	$\boxed{3}$

- Switching by MDI key operation

The display group can be sequentially switched by SHIFT + PAGE ⇩ (pressing PAGE DOWN while holding down SHIFT) or SHIFT + PAGE ⇧.

Number of CNC paths	Path	Display group number (Parameter No. 13131)	Display order number (Parameter No. 13132)	Screen display (The numbers in the illustration below represent path numbers.) (Example of SHIFT+PAGE DOWN)
3 paths	Path 1	1	1	1 ⇨ 2 ⇨ 3
	Path 2	2	1	
	Path 3	3	1	
	Path 1	1	2	2 1 ⇨ 3
	Path 2	1	1	
	Path 3	2	1	

- Path select signal

The path select signals are used to select a path for which LCD/MDI operation and display are to be performed.
A path to be selected is determined using a combination of four signals.
The following four signals (HEAD) are used:

HEAD <G0063.0>
HEAD2 <G0062.7>
HEAD3 <G0408.1>
HEAD4 <G0408.2>

The table below indicates the relationships between the combinations of the path select signals and selected paths.

Path select signal				Selected path
HEAD4 <G0408.2>	HEAD3 <G0408.1>	HEAD2 <G0062.7>	HEAD <G0063.0>	
0	0	0	0	Path 1
0	0	0	1	Path 2
0	0	1	0	Path 3
0	0	1	1	Path 4

Parameter

13131	Group number for simultaneous display of multiple paths

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 4

This parameter sets a group for simultaneous display on one screen in a multi-path system.
The paths set to belong to the same group are displayed on one screen.
If the values for all paths are set to 0, the simultaneous multi-path display function is disabled.

> **NOTE**
> When specifying groups, specify group numbers not less than 1 successively.
> On 8.4" and 10.4" display units, up to three paths can be specified for simultaneous display.
> On a 15" and 19" display units, up to four paths can be specified for simultaneous display.

13132	Simultaneous multi-path display order number

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of paths included in a simultaneous multi-path display group

This parameter sets the display order of a path defined to belong to a simultaneous multi-path display group.
Set the order, using numbers ranging from 1 to the number of paths included in the simultaneous multi-path display group.

[Example] Setting of simultaneous display group numbers and simultaneous display order numbers

Number of paths of CNC	Path	Display group number	Intra-group display order number	Screen display (Numbers represent displayed path numbers.)
One path	Path 1	1	1	1
Three path	Path 1	1	1	1 2 3
	Path 2	1	2	
	Path 3	1	3	
	Path 1	1	1	1 ⇨ 2 ⇨ 3
	Path 2	2	1	
	Path 3	3	1	
	Path 1	1	2	2 1 ⇨ 3
	Path 2	1	1	
	Path 3	2	1	

> **NOTE**
> Specify successive order numbers not less than 1 for the paths defined to belong to a group.

	#7	#6	#5	#4	#3	#2	#1	#0
3208			PSC					

[Input type] Setting input
[Data type] Bit

#5 PSC When the path is switched based on the path switch signal:
0: The screen display is switched to the last selected screen of the path.
1: The same screen as for the path before switching is displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
11304								PGR

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 PGR When the path select signal is changed, the screen of the multi path simultaneous display group:
0: Is not switched.
1: Is switched to the display group including the selected path.

> **NOTE**
> This parameter is initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

Signal

Path select signals

HEAD<G063.0>, HEAD2<G0062.7>, HEAD3<G0408.1>, HEAD4<G0408.2>

[Classification] Input signal
[Function] These signals select the target path for which MDI operation and display are to be performed.
Operations with the MDI panel are performed for a path specified by the combination of HEAD, HEAD2, HEAD3, and HEAD4.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0062	HEAD2							
G0063								HEAD
G0408						HEAD4	HEAD3	

13.3.2 Simultaneous Multi Path Program Editing

Overview

Simultaneous multi path program editing allows the user to edit programs for more than one path on a single screen according to the settings of the simultaneous multi path display parameters (parameters Nos.13131 and 13132).
This function is enabled by setting bit 0 (DHD) of parameter No. 3106.
For details of parameter Nos. 13131 and 13132, see Subsection "Multi Path Display".

Screen display

Fig. 13.3.2 (a) to Fig. 13.3.2 (c) show sample screens on which simultaneous multi path program editing is being performed.
The status line in the upper part of each program shows the name of the program, "FG-EDIT" indicating that foreground editing is in progress, and the path name.
The status line of the currently edited program is highlighted.

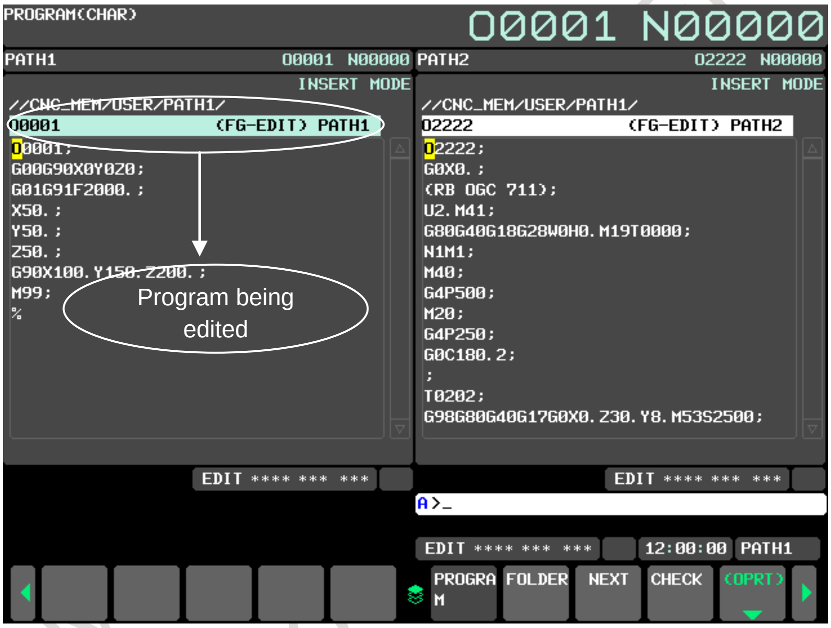


Fig. 13.3.2 (a) Simultaneous multi path program edit screen (10.4” display unit)

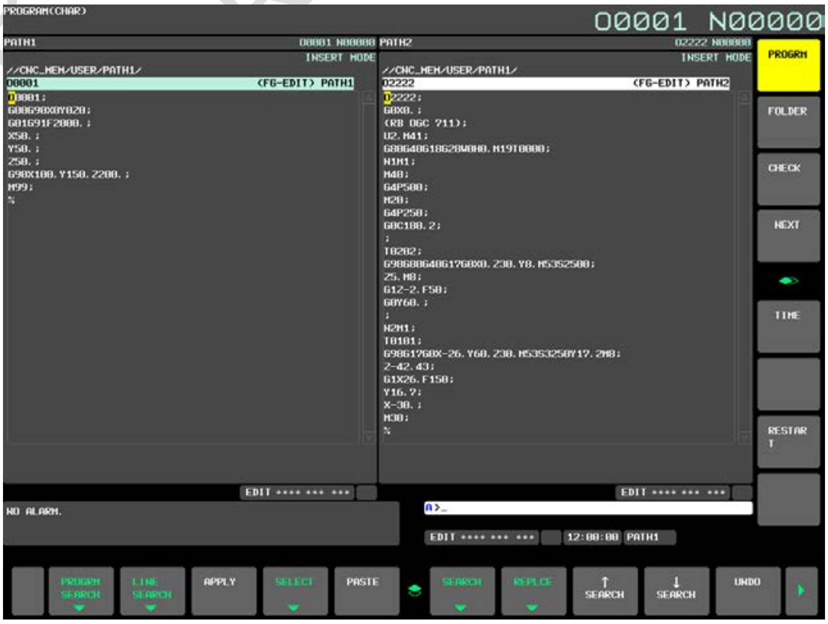


Fig. 13.3.2 (b) Simultaneous multi path program edit screen (15” display unit)

Explanation

- Mode

When the target paths for simultaneous display are in the EDIT mode or the MEM mode, the programs for these paths are displayed simultaneously on the program screen. When the EDIT mode is selected for a path, selecting that path allows program editing.
Fig. 13.3.2 (c) shows an example in which the EDIT mode and MEM mode are set together.
The first path is in the MEM mode (execution status screen), and the second path is in the EDIT mode (edit screen).
Soft keys are displayed according to the mod of the path currently selected.

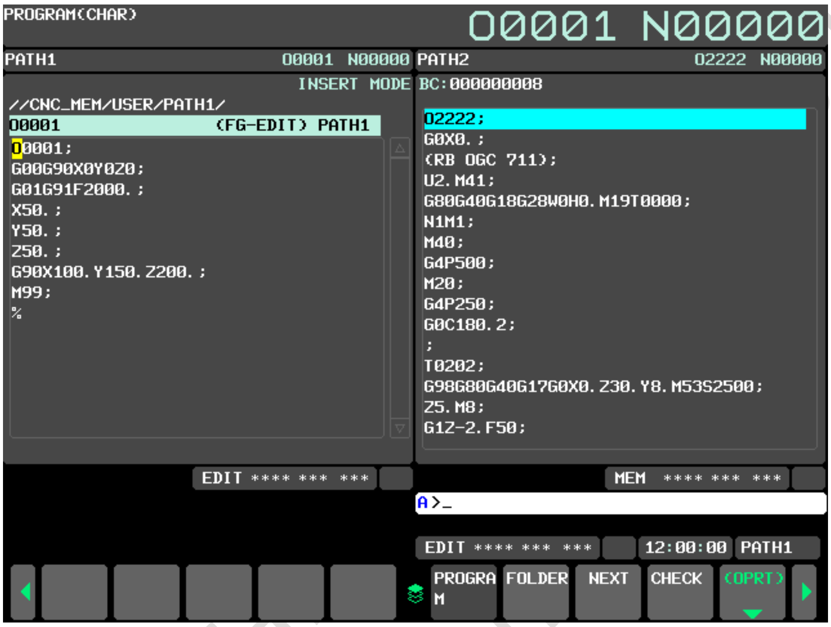


Fig. 13.3.2 (c) Screen showing the MEM mode and EDIT mode

- Changing the path to be edited

The path selected by the path selection signals is to be edited.

- Maximum number of paths for which simultaneous editing is enabled

The maximum number of paths for which simultaneous editing is enabled depends on the display unit as follows:

Display unit size	Maximum number of paths for simultaneous editing
8.4 and 10.4”	3 paths
15 and 19”	4 paths

- Conditions for disabling simultaneous editing

Simultaneous multi path program editing is disabled in the following cases:

- The program screen is displayed with the entire screen size.
- The paths selected for simultaneous display include a path that is in a mode other than the EDIT and MEM modes.
- The virtual MDI key function is enabled.

If background editing is started during simultaneous multi path program editing, background editing is performed with the entire screen size. Simultaneous multi path program editing and background editing cannot be performed at the same time.

- Simultaneous editing with a 8.4" display unit

When simultaneous editing is performed with a 8.4" display unit, displayed characters become smaller.
The number of characters displayed per path in the edit area is as follows:

- When simultaneous display is not performed, characters are displayed in 38 columns by 10 lines.
- For simultaneous display for one path, characters are displayed in 74 columns by 15 lines.
- For simultaneous display for two paths, characters are displayed in 35 columns by 14 lines.
- For simultaneous display for three paths, characters are displayed in 22 columns by 14 lines.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3106								DHD

[Input type] Setting input
[Data type] Bit

#0 DHD On the program screen:
0: Only a selected path can be edited and displayed.
1: Multiple paths can be edited and displayed at the same time.

13.4 HIGH-SPEED PROGRAM MANAGEMENT

Overview

When many kinds of machining is arranged, this function can achieve the short processing time when many programs are registered collectively or all programs are deleted. The program is not saved automatically to nonvolatile memory at registration, modification, or deletion of program.

Explanation

In conventional program management, the program is saved automatically to nonvolatile memory at registration, modification, or deletion of one program. Therefore, if many programs are registered at a time or all programs are deleted, all processing time becomes long, because automatic save is frequently done.

In this function, if bit 7 (HPM) of parameter No.11354 is set to 1, the processing time of registering many programs or deleting all programs can be shorten because automatic save to nonvolatile memory is invalid.

Perform saving operation by the all programs save request signal HPMRSV <G0514.4> or the application by FOCAS2/ C Language Library, if it is necessary that programs are saved to nonvolatile memory.

Function	Explanation
cnc_saveprog_start	Start saving program to nonvolatile memory.
cnc_saveprog_end	Check of end saving program to nonvolatile memory.

The status that programs aren't saved to nonvolatile memory can be confirmed by the programs not saved status signal HPMNTS <F0547.5>.
During the saving all programs to nonvolatile memory, the programs saving in progress signal HPMSVM <F0547.6> becomes "1".

> **NOTE**
> 1 There are no changes about the display of programs and the editing operation method of programs.

NOTE

2 In cases that this function is enabled, if the power supply is turned off without saving, the changed programs data is not saved.
The program selection by the following operation also is not saved.
- MDI operation
- External workpiece number search
- External program number search

Please perform saving operation by the signal or the application by FOCAS2/ C Language Library, if it is necessary to preserve the changed programs data/ the program selection after the power supply is turned off.

3 When the automatic operation signal is "1" in any path, the save operation is not executed.

4 When the programs not saved status signal HPMNTS is "0", if the all programs save request to nonvolatile memory start, the save operation is not executed because of unnecessary.

5 If the power supply is turned off during saving the programs programs (the programs saving in progress signal HPMSVM is "1"), all programs are deleted. In this case, an alarm PS0519 occurs when the power supply is turned on next.

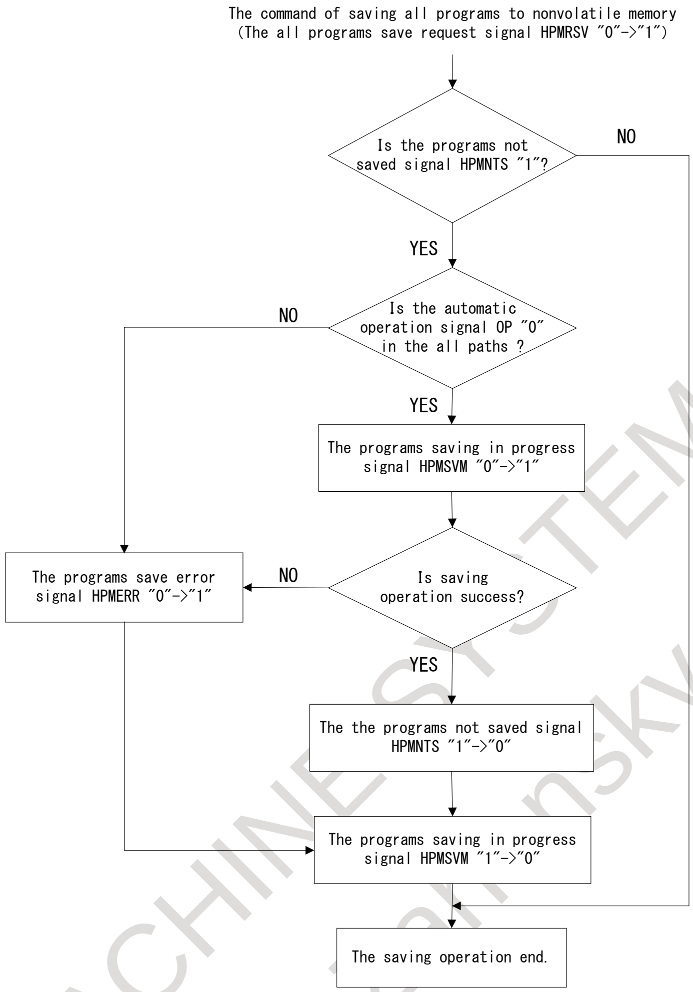
6 Do not save all programs to nonvolatile memory too frequently.

7 In automatic data backup function, if this function is enabled, bit 2 (AAP) of parameter No.10340 should be set to 0.

8 When the embedded macro program is registered, bit 7 (HPM) of parameter No.11354 should be set to 0.

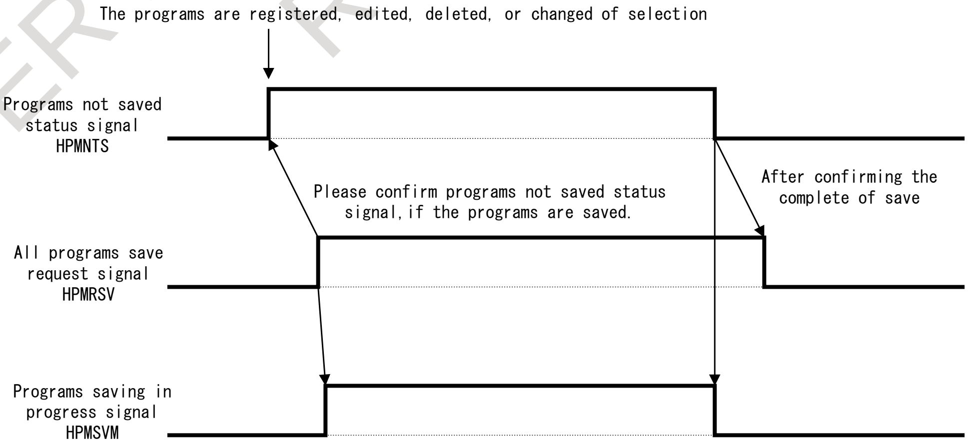
9 This function can be enabled only on CNC program storage memory (CNC_MEM device).

The flow chart of all programs saving to nonvolatile memory by the signal

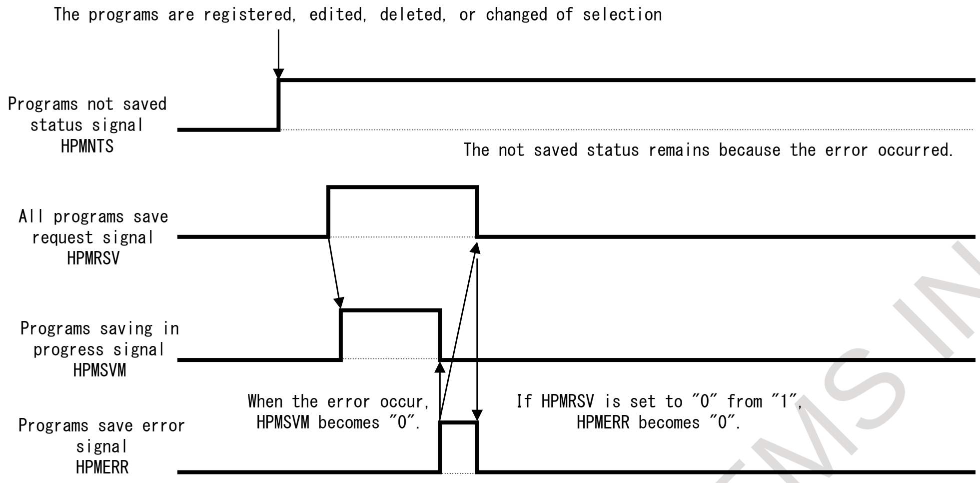


The time chart of all programs saving to nonvolatile memory by the signal

- The case of success



- The case of error



If the all programs save request signal HPMRSV is set to "1" from "0" in the automatic operation, the programs error signal becomes "1" at once.

Signal

All programs save request signal HPMRSV <G0514.4>

[Classification] Input signal

[Function] On high-speed program management, the save of all programs is requested by this signal.

[Operation] When the programs not saved status signal HPMNTS is "1", if this signal is set to "1" from "0", the save of all programs to nonvolatile memory start.
During program saving, the programs saving in progress signal HPMSVM is "1".
This signal is effective when bit 7 (HPM) of parameter No.11354 is "1".

NOTE
1 In the case of the following conditions, the save request does not start even if this signal is set to "1" from "0".
   - When the automatic operation signal is "1" in any path.
   - When the programs not saved status signal HPMNTS is "0".
   - When the programs saving in progress signal HPMSVM is "1".
   - When the automatic data backup executing signal ATBK is "1".
2 If the power supply is turned off during saving the programs (the programs saving in progress signal HPMSVM is "1"), all programs are deleted. In this case, an alarm PS0519 occurs when the power supply is turned on next. Therefore, please preserve the backup of the programs in the external memory beforehand.
3 Do not save all programs to nonvolatile memory too frequently.

Programs not saved status signal HPMNTS <F0547.5>

[Classification] Output signal

[Function] On high-speed program management, the signal indicates the signal indicates the status that the programs are not saved.

[Operation] When the programs are registered, edited, deleted, or changed of selection, this signal becomes "1".
If the programs are saved to nonvolatile memory by the all programs save request signal HPMRSV or the application by FOCAS2/ C Language Library, this signal becomes "0".
This signal is effective when bit 7 (HPM) of parameter No.11354 is "1".

Programs saving in progress signal HPMSVM <F0547.6>

[Classification] Output signal

[Function] On high-speed program management, the signal indicates the saving mode of the programs.

[Operation] During saving the programs to nonvolatile memory by the all programs save request signal HPMRSV or the application by FOCAS2/ C Language Library, this signal become "1". When the saving operation ends, this signal becomes "0".
This signal is effective when bit 7 (HPM) of parameter No.11354 is "1".

> **NOTE**
> 1 If the power supply is turned off when this signal is "1", all programs are deleted. In this case, an alarm PS0519 occurs when the power supply is turned on next.
> 2 If the time of saving is extremely short, this signal might not become "1".

Programs save error signal HPMERR <F0547.7>

[Classification] Output signal

[Function] On high-speed program management, the signal indicates the occurrence of the programs saving error.

[Operation] In the following condition when the programs saved by the all programs save request signal HPMRSV, this signal become "1" is requested, this signal becomes "1".
- When the automatic operation signal is "1" in either path.
- If the error is occurred, during saving to nonvolatile memory.

If all programs save request signal HPMRSV is set to "0" from "1", this signal becomes "0".
This signal is effective when bit 7 (HPM) of parameter No.11354 is "1".

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0514				HPMRSV				
F0547	HPMERR	HPMSVM	HPMNTS					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11354	HPM							

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#7 HPM High-speed program management is
0: Disabled.
1: Enabled.

NOTE

1 In cases that this function is enabled, if the power supply is turned off without saving, the changed programs data is not saved.
The program selection by the following operation also is not saved.
- MDI operation
- External workpiece number search
- External program number search

Please perform saving operation by the all programs save request signal HPMRSV or the application by FOCAS2/ C Language Library, if it is necessary to preserve the changed programs data/ the program selection after the power supply is turned off.

2 When the automatic operation signal is "1" in any path, the save operation is not executed.

3 Even if this parameter is changed from 1 to 0, automatic save of programs isn't executed until the power supply will be turned on next.

4 If the power supply is turned off during saving the programs (the programs saving in progress signal HPMSVM is "1"), all programs are deleted. In this case, an alarm PS0519 occurs when the power supply is turned on next.

5 Do not save all programs to nonvolatile memory too frequently.

6 In automatic data backup function, if this function is enabled, bit 2 (AAP) of parameter No.10340 should be set to 0.

7 When the embedded macro program is registered, bit 7 (HPM) of parameter No.11354 should be set to 0.

8 This function can be enabled only on CNC program storage memory (CNC_MEM device).

Alarm and message

No.	Message	Description
PS0519	PROGRAM FILES ARE BROKEN AND CLEARED	The program have been cleared, because the program were broken. The power supply might have been turned off while the program files is being saved to nonvolatile memory.

14 INPUT/OUTPUT OF DATA

14.1 RS-232C INTERFACE

Overview

The data shown below can be input/output through RS-232C interface.

1. Program
2. Offset data
3. Parameter
4. Pitch error compensation data
5. Custom macro common variables.
6. Workpiece coordinate system setting data
7. Operation history data (output only)
8. Tool management data

The above data can be output to a memory card or USB memory via a memory card interface or USB memory interface.

When a memory card is selected as the data input/output destination (parameter No. 20 = 4) at this time, if a file with the same file name already exists during writing of NC data such as programs or parameters, it is possible to select whether to overwrite the file or cancel the operation.
To enable this function, set bit 1 (COW) of parameter No. 11308.

When a USB memory is selected as the data input/output destination (parameter No. 20 = 17), a file with the same file name is already present during writing of NC data, the file is overwritten.

The channel for data I/O is determined by setting parameter No. 0020. In this case, foreground or background data I/O is restricted to one channel. In addition, data I/O can be controlled separately by setting bit 0 (IO4) of parameter No. 0110. Concretely, a channel can be assigned to each of foreground input, foreground output, background input, and background output.

Explanation

The parameters described below must be set up to use an I/O unit interface (RS-232C serial port) or memory card interface for inputting and outputting data (such as programs and parameters) between external input/output units and memory cards.
The channel to which an I/O unit is connected needs be specified by setting I/O CHANNEL (parameter No. 0020). In addition, the specification number, baud rate, and number of stop bits of each I/O unit must be set in the parameter corresponding to each channel in advance. For setting of channel 1, two types of parameters for setting I/O units are provided.
Channel setting can be made for each of foreground I/O and background I/O by setting bit 0 (IO4) of parameter No. 0110. At this time, channel setting is made with parameters Nos. 0020, 0021, 0022, and 0023.
The following shows correlation of interface parameters related to individual channels.

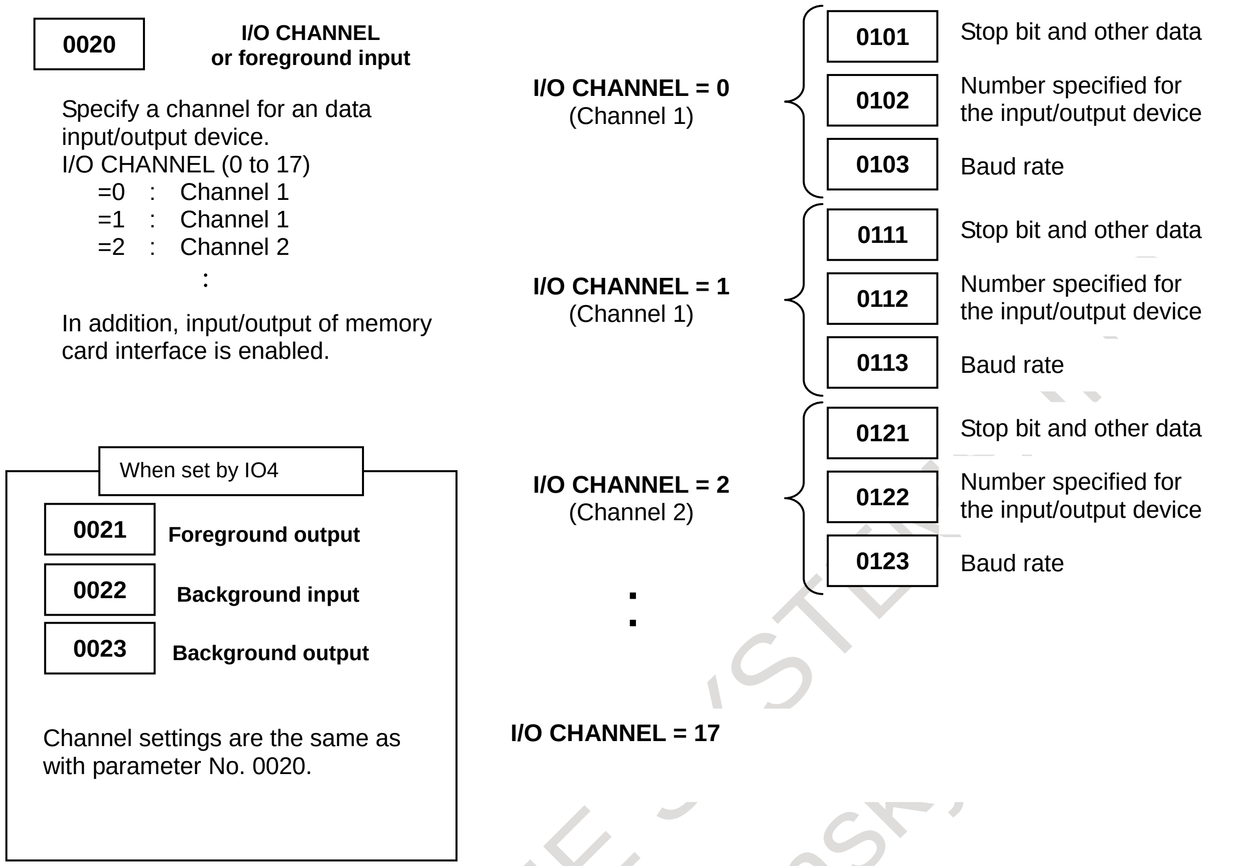


Fig. 14.1 (a) Correlation of interface parameters related to individual channels

Diagnosis

- Foreground

030	CHARACTER NUMBER DATA

When the alarm SR0001, “TH ERROR” occurred, the position in which it occurred is indicated by the number of characters counted from the beginning of the block.

031	TH DATA

Readout code of the number of characters in which an alarm SR0001, “TH ERROR” occurred

- Background

032	CHARACTER NUMBER DATA

When the alarm SR0001, “TH ERROR” occurred, the position in which it occurred is indicated by the number of characters counted from the beginning of the block.

033	TH DATA

Readout code of the number of characters in which an alarm SR0001, “TH ERROR” occurred

Parameter

0020	I/O CHANNEL : Input/output device selection, or interface number for a foreground input device
0021	Foreground output device setting
0022	Background input device setting
0023	Background output device setting

[Input type] Setting input
[Data type] Byte
[Valid data range] 0 to 17

The CNC has the following interfaces for transferring data to and from an external input/output device and the host computer:

- Input/output device interface (RS-232C serial ports 1 and 2)
- Memory card interface
- Data server interface
- Embedded Ethernet interface
- USB memory interface

By setting bit 0 (IO4) of parameter No. 0110, data input/output can be controlled separately. When IO4 is not set, data input/output is performed using the channel set in parameter No. 0020. When IO4 is set, a channel can be assigned to each of foreground input, foreground output, background input, and background output.

In these parameters, specify the interface connected to each input/output device to and from which data is to be transferred. See the table below for these settings.

Correspondence between settings and input/output devices

Setting	Description
0,1	RS-232C serial port 1
2	RS-232C serial port 2
4	Memory card interface
5	Data server interface
9	Embedded Ethernet interface
17	USB memory interface

- Parameters Common to all Channels

	#7	#6	#5	#4	#3	#2	#1	#0
0000							ISO	TVC

[Input type] Setting input
[Data type] Bit path

#0 TVC TV check
0: Not performed
1: Performed

#1 ISO Code used for data output
0: EIA code
1: ISO code

NOTE
1. The I/O setting of a memory card is made by bit 0 (ISO) of parameter No. 0139.
2. The I/O setting of an USB memory is made by bit 0 (ISU) of parameter No. 11505.
3. The I/O setting of a data server is made by bit 0 (ISO) of parameter No. 0908.
4. When EIA code is used for data output (ISO = 0), set bit 3 (ASI) of parameter No.101 and 111 and 121 to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
0010						PEC	PRM	PZS

[Input type] Setting input
[Data type] Bit path

#0 PZS When a part program is output, the O number is:
0: Not zero-suppressed.
1: Zero-suppressed.

#1 PRM When parameters are output, the parameters whose values are 0 are:
0: Output.
1: Not output.

#2 PEC When pitch error compensation data is output, the data whose value is 0 is:
0: Output.
1: Not output.

	#7	#6	#5	#4	#3	#2	#1	#0
0100	ENS	IOP			NCR	CRF	CTV	

[Input type] Setting input
[Data type] Bit

#1 CTV Character counting for TV check in the comment section of a program.
0: Performed
1: Not performed

#2 CRF Output of the end of block (EOB) in ISO code
0: Depends on the setting of bit 3 (NCR) of parameter No. 0100.
1: CR, LF are output.

#3 NCR Output of the end of block (EOB) in ISO code
0: LF, CR, CR are output.
1: Only LF is output.

#6 IOP Stopping a program output or input operation by a reset is:
0: Enabled
1: Disabled

#7 ENS Action taken when a NULL code is found during read of EIA code
0: An alarm is generated.
1: The NULL code is ignored.

		#7	#6	#5	#4	#3	#2	#1	#0
0110									IO4

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 IO4 Separate control of I/O channel numbers is:
0: Not performed.
1: Performed.
If the I/O channels are not separately controlled, set the input/output device in parameter No. 0020.
If the I/O channels are separately controlled, set the input device and output device in the foreground and the input device and output device in the background in parameters Nos. 0020 to 0023 respectively.
Separate control of I/O channels makes it possible to perform background editing, program input/output, and the like during the DNC operation.

		#7	#6	#5	#4	#3	#2	#1	#0
0139									ISO

[Input type] Setting input
[Data type] Bit

#0 ISO When a memory card is selected as an I/O device, data input/output is performed using
0: ASCII codes.
1: ISO codes.

> ⚠ **WARNING**
> 1 Unless data is input using ASCII codes, set this parameter to 1 to input or output data using ISO codes.
> 2 Data input/output with ASCII codes is dangerous because parity information is not included and a data error during the data input/output is not detected.
> 3 DNC operation from a memory card also must set the parameter to 1, and execute DNC operation by ISO code. ASCII code is dangerous because parity information is not included and a data error during the data input is not detected.

		#7	#6	#5	#4	#3	#2	#1	#0
0908									ISO

[Input type] Setting input
[Data type] Bit

#0 ISO When a data server is selected as an I/O device, data input/output is performed using
0: ASCII codes.
1: ISO codes.

> ⚠ **WARNING**
> 1 Unless data is input using ASCII codes, set this parameter to 1 to input or output data using ISO codes.
> 2 Data input/output with ASCII codes is dangerous because parity information is not included and a data error during the data input/output is not detected.
> 3 DNC operation from a data server also must set the parameter to 1, and execute DNC operation by ISO code. ASCII code is dangerous because parity information is not included and a data error during the data input is not detected.

	#7	#6	#5	#4	#3	#2	#1	#0
11308							COW	

[Input type] Parameter input
[Data type] Bit

#1 COW When the file of specified name already exists on memory card,
0: It is not overwritten
Alarm SR1973, "FILE ALREADY EXIST" is generated.
1: It is overwritten.
Because the confirmation message is displayed before overwriting even if COW is 1, overwriting can be canceled.

> **NOTE**
> When the overwritten file is read only attribute, it is not possible to overwrite even if bit 1 (COW) of parameter No. 11308 = 1.

	#7	#6	#5	#4	#3	#2	#1	#0
11505								ISU

[Input type] Setting input
[Data type] Bit

#0 ISU When an USB memory card is selected as an I/O device, data input/output is performed using
0: ASCII codes.
1: ISO codes.

> ⚠ **WARNING**
> 1 Unless data is input using ASCII codes, set this parameter to 1 to input or output data using ISO codes.
> 2 Data input/output with ASCII codes is dangerous because parity information is not included and a data error during the data input/output is not detected.

- Parameters of Channel 1 (I/O CHANNEL=0)

	#7	#6	#5	#4	#3	#2	#1	#0
0101	NFD				ASI			SB2

[Input type] Parameter input
[Data type] Bit

#0 SB2 The number of stop bits
0: 1
1: 2

#3 ASI Code used at data input/output
0: EIA or ISO code (input: automatically distinguished/output: decided by the setting parameter of bit 1 (ISO) of parameter No.0000)
1: ASCII code

> **NOTE**
> When ASCII code is used for data input/output (ASI = 1), set bit 1 (ISO) of parameter No.0000 to 1.

#7 NFD Feed before and after the data at data output
0: Output
1: Not output
When input/output devices other than the FANUC PPR are used, set NFD to 1.

0102	Number specified for the input/output device (when the I/O CHANNEL is set to 0)

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 6
Set the specification number of the input/output device corresponding to I/O CHANNEL=0.
The following table lists the specification numbers and corresponding input/output device specifications.

Specification numbers and corresponding input/output device specifications

Specification number	Input/output device specification
0	RS-232C (control codes DC1 to DC4 are used)
1	FANUC CASSETTE ADAPTOR 1(FANUC CASSETTE B1/B2)
2	FANUC CASSETTE ADAPTOR 3(FANUC CASSETTE F1)
3	FANUC PROGRAM FILE Mate、FANUC FA Card Adaptor, FANUC FLOPPY CASSETTE ADAPTOR, FANUC Handy File FANUC SYSTEM P-MODEL H
4	RS-232C (control codes DC1 to DC4 are not used)
5	Portable tape reader
6	FANUC PPR FANUC SYSTEM P-MODEL G, FANUC SYSTEM P-MODEL H

0103	Baud rate (when I/O CHANNEL is set to 0)

[Input type] Parameter input
[Data type] Byte
[Valid data range] 1 to 12

Set the baud rate of the input/output device corresponding to I/O CHANNEL=0.
When setting this parameter, see the following table:

Baud rates and corresponding settings

Setting	Baud rate (bps)	Setting	Baud rate (bps)
1	50	8	1200
3	110	9	2400
4	150	10	4800
6	300	11	9600
7	600	12	19200

- Parameters of Channel 1 (I/O CHANNEL=1)

	#7	#6	#5	#4	#3	#2	#1	#0
0111	NFD				ASI			SB2

[Input type] Parameter input
[Data type] Bit

#0 SB2 The number of stop bits
0: 1
1: 2

#3 ASI Code used at data input/output
0: EIA or ISO code (input: automatically distinguished/output: decided by the setting parameter of bit 1 (ISO) of parameter No.0000)
1: ASCII code

> **NOTE**
> When ASCII code is used for data input/output (ASI = 1), set bit 1 (ISO) of parameter No.0000 to 1.

#7 NFD Feed before and after the data at data output
0: Output
1: Not output

When input/output devices other than the FANUC PPR are used, set NFD to 1.

0112	Number specified for the input/output device (when the I/O CHANNEL is set to 1)

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 6

Set the specification number of the input/output device corresponding to I/O CHANNEL=1.

0113	Baud rate (when I/O CHANNEL is set to 1)

[Input type] Parameter input
[Data type] Byte
[Valid data range] 1 to 12
Set the baud rate of the input/output device corresponding to I/O CHANNEL=1.

- Parameters of Channel 2 (I/O CHANNEL=2)

	#7	#6	#5	#4	#3	#2	#1	#0
0121	NFD				ASI			SB2

[Input type] Parameter input
[Data type] Bit

#0 SB2 The number of stop bits
0: 1
1: 2

#3 ASI Code used at data input/output
0: EIA or ISO code (input: automatically distinguished/output: decided by the setting parameter of bit 1 (ISO) of parameter No.0000)
1: ASCII code

> **NOTE**
> When ASCII code is used for data input/output (ASI = 1), set bit 1 (ISO) of parameter No.0000 to 1.

#7 NFD Feed before and after the data at data output
0: Output
1: Not output

0122	Number specified for the input/output device (when the I/O CHANNEL is set to 2)

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 6
Set the specification number of the input/output device corresponding to I/O CHANNEL=2.

0123	Baud rate (when I/O CHANNEL is set to 2)

[Input type] Parameter input
[Data type] Byte
[Valid data range] 1 to 12
Set the baud rate of the input/output device corresponding to I/O CHANNEL=2.

- Parameters of DNC operation from the memory card

	#7	#6	#5	#4	#3	#2	#1	#0
0138	**MNC**		**SCH**					

[Input type] Parameter input
[Data type] Bit

#5 SCH Schedule operation function is:
0: Disabled.
1: Enabled.

#7 MNC DNC operation from the memory card and external device subprogram call from the memory card are:
0: Not performed.
1: Performed.

Alarm and message

Number	Message	Description
SR0001	TH ERROR	A TH error was detected during reading from an input device. The read code that caused the TH error and how many statements it is from the block can be verified in the diagnostics screen.
SR0002	TV ERROR	An error was detected during the single–block TV error. The TV check can be suppressed by setting bit 0 (TVC) of parameter No. 0000 to 0.
SR1805	ILLEGAL COMMAND	An attempt was made to specify an illegal command during I/O processing on an I/O device.
SR1806	DEVICE TYPE MISS MATCH	An operation not possible on the I/O device that is currently selected in the setting was specified. This alarm is also generated when file rewind is instructed even though the I/O device is not a FANUC Cassette.
SR1807	PARAMETER SETTING ERROR	An I/O interface option that has not yet been added on was specified. The external I/O device and baud rate, stop bit and protocol selection settings are erroneous.
SR1808	DEVICE DOUBLE OPENED	An attempt was made to open a device that is being accessed.
SR1973	FILE ALREADY EXIST	A file having the same name already exists on the memory card.

- Channel 1

Number	Message	Description
SR0085	OVERRUN ERROR	During a read by RS-232C interface 1, an overrun, parity, or framing error occurred. The number of bits for entered data was not matched, or baud rate setting or I/O device specification number was incorrect.
SR0086	DR OFF	During I/O process by RS-232C interface 1, the data set ready input signal of the I/O device (DR) was OFF. Possible causes are an I/O device not turn on, a broken cable, and a defective printed circuit board.
SR0087	BUFFER OVERFLOW	During a read by RS-232C interface 1, although a read stop command was issued, more than 10 characters were input. The I/O device or printed circuit board was defective.

- Channel 2

Number	Message	Description
SR1830	DR OFF(2)	During I/O process by RS-232C interface 2, the data set ready input signal of the I/O device (DR) was OFF. Possible causes are an I/O device not turn on, a broken cable, and a defective printed circuit board.
SR1832	OVERRUN ERROR(2)	During a read by RS-232C interface 2, an overrun, parity, or framing error occurred. The number of bits for entered data was not matched, or baud rate setting or I/O device specification number was incorrect.
SR1834	BUFFER OVERFLOW(2)	During a read by RS-232C interface 2, although a read stop command was issued, more than 10 characters were input. The I/O device or printed circuit board was defective.

- DNC operation

Number	Message	Description
PS0123	ILLEGAL MODE FOR GOTO/WHILE/DO	A GOTO statement or WHILE–DO statement was found in the program in the MDI or DNC mode. Modify the program.
PS1081	EXT DEVICE SUB PROGRAM CALL MODE ERROR	1. M198 or M99 was executed during scheduled operation. Alternatively, M198 was executed during DNC operation. Modify the program. 2. An interruption macro was specified and M99 was executed during pocketing of multiple repetitive canned cycles.

14.2 NC DATA OUTPUT FUNCTION

Overview

This function outputs the following various data into memory card or USB memory by one operation.

- SRAM data
- User files (made by user such as PMC ladder)
- Text data (Parameter, program, and so on)

This function enables to back up the system data easily.
Moreover, the contents of SRAM data can be confirmed immediately because both SRAM data and text data are output.
NC data output signal ALLO<F0578.5> is set to "1" from the beginning to the end of outputting text data.

Signal

NC data output signal ALLO<G0578.5>

[Classification] Output signal

[Function] This signal indicates that text data are under output by NC data output function.

[Output condition] This signal becomes "1", when:

- Outputting text data by NC data output function is started.

This signal becomes "0", when:

- Outputting text data by NC data output function is finished.
- Outputting text data by NC data output function is terminated abnormally.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn578			ALLO					

Parameter

0020	**I/O CHANNEL : Input/output device selection, or interface number for a foreground input device**
0021	**Foreground output device setting**
0022	**Background input device setting**
0023	**Background output device setting**

[Input type] Setting input
[Data type] Byte
[Valid data range] 0 to 17

The CNC has the following interfaces for transferring data to and from an external input/output device and the host computer:

- Input/output device interface (RS-232C serial ports 1 and 2)
- Memory card interface
- Data server interface
- Embedded Ethernet interface
- USB memory interface

By setting bit 0 (IO4) of parameter No. 0110, data input/output can be controlled separately. When IO4 is not set, data input/output is performed using the channel set in parameter No. 0020. When IO4 is set, a channel can be assigned to each of foreground input, foreground output, background input, and background output.
In these parameters, specify the interface connected to each input/output device to and from which data is to be transferred. See the table below for these settings.

Correspondence between settings and input/output devices

Setting	Description
0,1	RS-232C serial port 1
2	RS-232C serial port 2
4	Memory card interface of CNC Memory card/USB interface in the secondary display unit for Ethernet connection or the shared display unit for Ethernet connection
5	Data server interface
9	Embedded Ethernet interface
17	USB memory interface

	#7	#6	#5	#4	#3	#2	#1	#0
0110								IO4

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 IO4 Separate control of I/O channel numbers is:
0: Not performed.
1: Performed.
If the I/O channels are not separately controlled, set the input/output device in parameter No. 0020.
If the I/O channels are separately controlled, set the input device and output device in the foreground and the input device and output device in the background in parameters No. 0020 to No. 0023 respectively.
Separate control of I/O channels makes it possible to perform background editing, program input/output, and the like during the DNC operation.

	#7	#6	#5	#4	#3	#2	#1	#0
0138								MDP

[Input type] Parameter input
[Data type] Bit

#0 MDP To the extensions of input/output files, a path number is:
0: Not added.
1: Added.

> **NOTE**
> If a file name is specified by setting F, this parameter is ignored, and a path number is not added to the extension.

	#7	#6	#5	#4	#3	#2	#1	#0
0313							TFO	BOP

[Input type] Parameter input
[Data type] Bit

#0 BOP NC data output function is:
0: Disabled.
1: Enabled.

> **NOTE**
> If a file with the same name already exists in the external I/O device, the file is overwritten. It is recommended to clear the external I/O device before using this function.

#1 TFO On NC data output function, text data (such as parameter, program) is:
0: Output.
1: Not output.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	NC data output function

15 MEASUREMENT

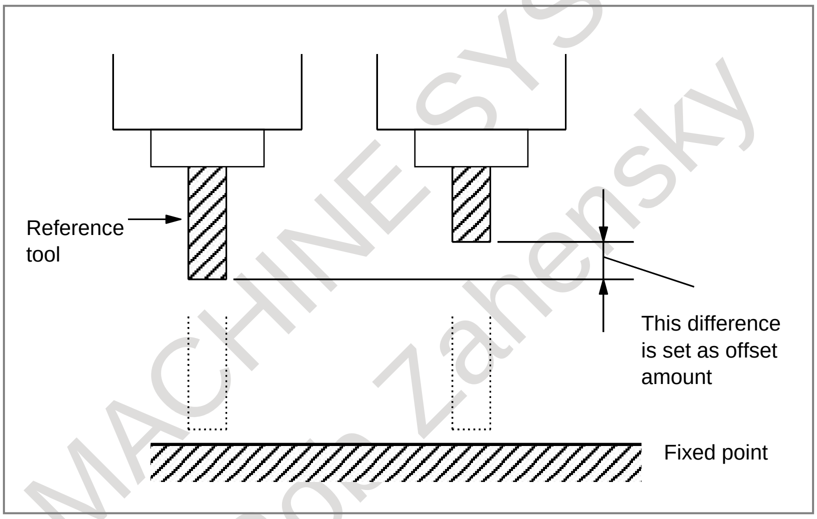
15.1 TOOL LENGTH MEASUREMENT

M

Overview

The value displayed as a relative and machine position can be set in the offset memory as an offset value by a soft key.

Switch to the offset value display screen on the screen. Relative positions are also displayed on this screen. Then select the reference tool and set it at the fixed point on the machine by manual operation. Reset the displayed relative position to zero. Set the tool for measurement at the same fixed point on the machine by manual operation. The relative position display at this point shows difference between the reference tool and the tool measured and the relative position display value is then set as offset amounts. Another way is to display machine positions and compensate by inputting adjustment values.



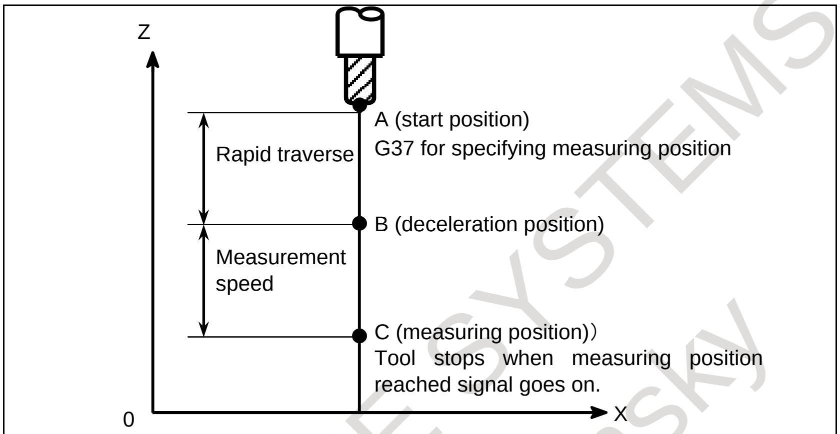
Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Tool length compensation

15.2 AUTOMATIC TOOL LENGTH MEASUREMENT (M SERIES) / AUTOMATIC TOOL OFFSET (T SERIES)

Overview

When a tool is moved to the measurement position by execution of a command given to the CNC, the CNC automatically measures the difference between the current coordinate value and the coordinate value of the command measurement position and uses it as the offset value for the tool. When the tool has been already offset, it is moved to the measurement position with that offset value.
The difference between the coordinate value of a measuring position and a specified coordinate value is added to current compensation.



Compensation = (Current compensation) + [(Coordinate at tool stop) - (Coordinate at specification of measuring position)]

Format

M

Hxx ;	Specifies an offset number for tool length compensation.
G90 G37 IP_ ;	Automatic tool length measurement. Absolute programming. G37 is valid only in the block in which it is specified. IP_ indicates the X-, Y-, Z-, or fourth axis.

T

Txx ;	Specifies an offset number for tool position compensation.
G36 X_ ; or G37 Z_ ;	Automatic tool offset. Absolute programming. G36 and G37 are valid only in the block in which it is specified.

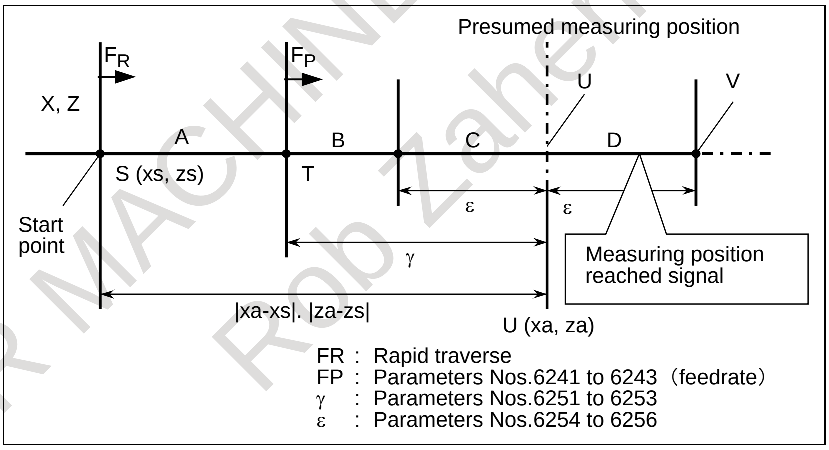
Signal

Measuring position reached signals

XAE1#1<X0004.0>, XAE2#1<X0004.1> (M/T series), XAE3#1<X0004.2> (M series only)
XAE1#2<X0013.0>, XAE2#2<X0013.1> (M/T series), XAE3#2<X0013.2> (M series only)
XAE1#3<X0011.0>, XAE2#3<X0011.1> (M/T series), XAE3#3<X0011.2> (M series only)
GAE1<Gn517.0>,GAE2<Gn517.1>, GAE3<Gn517.2> (M/T series)

[Classification] Input signal

[Function] If the measuring position specified by a program command differs from the measuring position which a tool has reached in practice, that is, the position at the moment the measuring position reached signal has just been turned "1", the difference in the coordinate value is added to the current tool compensation value to update the compensation value. The tool is first fed to the specified measuring position by rapid traverse in a block where one of the following commands has been specified:
G37 (Mseries)
G36,G37 (Tseries) (G37.1, G37.2 if the bit 6 (MDC) of parameter No. 6210 is set to 1)
The tool decelerates and temporarily stops at the distance γ before the measuring position. The tool then moves to the measuring position at the speed preset by a parameters Nos. 6241 to 6243. If the measuring position reached signal corresponding to the G code is turned "1" after the tool has approached within distance ε of the measuring position and before the tool overshoots the measuring position by distance ε, the control unit updates the compensation value and terminates the move command for the block.
If the measuring position reached signal is not turned "1" even after the tool has overshot the measuring position by distance ε, the control unit enters an alarm state and terminates the move command for the block without updating the compensation value.



Command code	Axis specification		Signal input		Valid parameters	
	T series	M series	T series	M series	T series	M series
G36 (bit 3 (G36) of parameter No. 3405=0)	Basic 1st axis		XAE1 or GAE1		6241, 6251, 6254	
G37 (bit 3 (G36) of parameter No. 3405=0)	Basic 3rd axis	Basic1st to 3rd axes	XAE2 or GAE2	XAE1 to XAE3 or GAE1 to GAE3	6242, 6252, 6255	6241 to 6243, 6251 to 6253, 6254 to 6256
G37.1 (bit 3 (G36) of parameter No. 3405=1)	Basic 1st axis	Basic 1st axis	XAE1 or GAE1	XAE1 or GAE1	6241, 6251, 6254	6241, 6251, 6254
G37.2 (bit 3 (G36) of parameter No. 3405=1)	Basic 3rd axis	Basic 2nd axis	XAE2 or GAE2	XAE2 or GAE2	6242, 6252, 6255	6242, 6252, 6255
G37.3 (bit 3 (G36) of parameter No. 3405=1)		Basic 3rd axis		XAE3 or GAE3		6243, 6253, 6256

NOTE

If parameters Nos. 6242 and 6243 are set to 0, the setting of parameter No. 6241 becomes valid.

[Operation] When the measuring position reached signals (XAE1-XAE3 or GAE1-GAE3) is turned "1", the control unit works as follows:

- Reads the position of the tool along the axis currently specified and updates the current compensation value based on the difference between the specified measuring position and the read measuring position in the following case: When the measuring position reached signal corresponding to the G code is turned on in a block where G36 (T series) or G37 is specified after the tool is within distance ε of the measuring position specified by a program and before the tool overshoots the measuring position by distance ε. The control unit then stops the tool, and terminates the move command for the block.
- Enters an alarm (PS0080) "G37 MEASURING POSITION REACHED SIGNAL IS NOT PROPERLY INPUT" state and terminates the move command for the block without updating the compensation value in the following case: When the measuring position reached signal corresponding to the command is turned "1" in a block where G36 (T series), G37 is specified after the tool is within distance γ of the measuring position but before the tool is within distance ε of the measuring position.
- The measuring position reached signal is not monitored on the rising edge, but the state of the signal is simply monitored. So, if the measuring position reached signal remains to be set to "1", and automatic tool length measurement (M series) (automatic tool offset (T series) (G36, G37) is then specified, the CNC issues the PS0080 alarm when a movement has been made to distance γ before the measurement position.

NOTE

1 The measuring position reached signal requires at least 10 msec.
2 A delay or variation in detecting a measuring position reached signal is 0 to 2 ms just on the CNC side, excluding the PMC side. Accordingly, the measurement error is the sum of 2 ms and a delay or variation in transferring a measuring position reached signal on the PMC side (including receiver delay or variation), multiplied by the feedrate specified in parameter No. 6241.
3 A delay or variation after the detection of a measuring position reached signal until feed stop is 0 to 8 ms. When an overshoot is calculated, an acceleration/deceleration delay, servo delay, PMC delay must also be considered.
4 Measuring position reached signals XAE1 to XAE3 need not be processed on the PMC side because the CNC reads the signals directly from the machine.
5 If the tool length measurement function is not used, the PMC can use the pins corresponding to the measuring position reached signals for general-purpose input signals.
6 When the bit 2 (XSG) of parameter No. 3008 is set to 1, address <X0004> including measuring position reached signals can be assigned to an X address specified in parameter No. 3019.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
X0004							XAE2[#1]	XAE1[#1]	(T series)
						XAE3[#1]	XAE2[#1]	XAE1[#1]	(M series)
X0013							XAE2[#2]	XAE1[#2]	(T series)
						XAE3[#2]	XAE2[#2]	XAE1[#2]	(M series)
X0011							XAE2[#3]	XAE1[#3]	(T series)
						XAE3[#3]	XAE2[#3]	XAE1[#3]	(M series)
Gn517						GAE3[#P]	GAE2[#P]	GAE1[#P]	

NOTE

To use independent measuring position reached signals for all the paths in a system having four or more paths, use a G address.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3008						XSG		

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#2 XSG A signal assigned to an X address is:
0: Fixed at the address.
1: Able to be reassigned to an arbitrary X address.

> **NOTE**
> When this parameter is set to 1, set parameters Nos. 3013, 3014, 3012, and 3019. If parameter No. 3013 and No. 3014 are not set, the deceleration signal for reference position return is assigned to bit 0 of <X0000>. If parameters Nos. 3012 and 3019 are not set, the skip signal, the PMC axis control skip signal, the measuring position reached signal, the interlock signal for each axis direction, and the tool compensation value write signal are assigned to <X0000>.

> ⚠ **WARNING**
> Please set an appropriate value to parameters (Nos. 3013, 3014, 3012, and 3019) beforehand when you set 1 to this parameter.
> If you used the set address in the parameters (Nos. 3013, 3014, 3012, and 3019) by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used X address.

3019	Address to which the PMC axis control skip signal, measuring position reached signal, and tool offset write signals are assigned

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 727

Set an X address to which the PMC axis control skip signal ESKIP, measuring position reached signals (XAE, YAE, and ZAE (M series) or XAE and ZAE (T series)), and tool offset write signals (±MIT1 and ±MIT2 (T series)) are to be assigned.

> ⚠ **WARNING**
> If you used the set address in this parameter by another usage, it may cause an unexpected machine behavior.
> Be careful enough that you must prevent the competition of the used X address.

	#7	#6	#5	#4	#3	#2	#1	#0
3405					G36			

[Input type] Parameter input
[Data type] Bit path

#3 G36 As a G code to be used with the automatic tool length measurement function (M series)/automatic tool offset function (T series) is:
0: G36 (T series only)/G37 is used.
1: G37.1/G37.2/G37.3 is used.

> **NOTE**
> If it is necessary to perform circular threading (counterclockwise) in T series, set this parameter to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
6201							SEB	

[Input type] Parameter input
[Data type] Bit path

#1 SEB When a skip signal, high-speed skip signal or measuring position reached signal goes on while the skip function, the continuous high-speed skip function or the automatic tool length measurement (M series) or automatic tool compensation (T series) is used, the accumulated pulses and positional deviation due to acceleration/deceleration are:
0: Ignored.
1: Considered and compensated.
The accumulated pulses and positional deviation due to actual acceleration/deceleration when the skip signal, high-speed skip signal or measuring position reached signal goes on are considered to obtain the position at which the signal is input.

	#7	#6	#5	#4	#3	#2	#1	#0
6210		MDC						
	CCM	MDC						

[Input type] Parameter input
[Data type] Bit path

#6 MDC The measurement result of automatic tool length measurement (M series) or automatic tool compensation (T series) is:
0: Added to the current offset.
1: Subtracted from the current offset.

#7 CCM The current offset amount of automatic tool length measurement (M series) is:
0: The offset amount set to the offset screen.
In case of tool offset memory C, the value for tool wear offset is selected.
1: The offset amount actually effected.

	#7	#6	#5	#4	#3	#2	#1	#0
6240								**AE0**

[Input type] Parameter input
[Data type] Bit path

#0 AE0 Measurement position arrival is assumed when the measuring position reached signals XAE1, XAE2, GAE1<Gn517.0> and GAE2<Gn517.1> (T series) or the measuring position reached signals XAE1, XAE2, XAE3, GAE1<Gn517.0>, GAE2<Gn517.1> and GAE3<Gn517.2> (M series) are:
0: 1.
1: 0.

> **NOTE**
> Setting this parameter to 1, input measuring position reached signals to both XAE1, XAE2, XAE3 and GAE1, GAE2, GAE3<Gn517.0, Gn517.1, Gn517.2>. If the other measuring position reached signals are not input, the CNC issues the PS0080 alarm to the movement to the position of the distance γ.

6241	**Feedrate during measurement of automatic tool compensation (T series) (for the XAE1 and GAE1 signals)**
	Feedrate during measurement of automatic tool length measurement (M series) (for the XAE1 and GAE1 signals)
6242	**Feedrate during measurement of automatic tool compensation (T series) (for the XAE2 and GAE2 signals)**
	Feedrate during measurement of automatic tool length measurement (M series) (for the XAE2 and GAE2 signals)
6243	
	Feedrate during measurement of automatic tool length measurement (M series) (for the XAE3 and GAE3 signals)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
These parameters set the relevant feedrate during measurement of automatic tool compensation (T series) or automatic tool length measurement (M series).

> **NOTE**
> When the setting of parameter No. 6242 or 6243 is 0, the setting of parameter No. 6241 is used.

6251	γ value on the X axis during automatic tool compensation (T series)
	γ value during automatic tool length measurement (M series) (for the XAE1 and GAE1 signals)

6252	γ value on the Z axis during automatic tool compensation (T series)
	γ value during automatic tool length measurement (M series) (for the XAE2 and GAE2 signals)

6253	
	γ value during automatic tool length measurement (M series) (for the XAE3 and GAE3 signals)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

These parameters set the relevant γ value during automatic tool compensation (T series) or automatic tool length measurement (M series).

NOTE
When the Reference axis (parameter No.1031) is Diameter specification, specify the diameter value. When the Reference axis (parameter No.1031) is Radius specification, specify the radius value.

6254	ε value on the X axis during automatic tool compensation (T series)
	ε value during automatic tool length measurement (M series) (for the XAE1 and GAE1 signals)

6255	ε value on the Z axis during automatic tool compensation (T series)
	ε value during automatic tool length measurement (M series) (for the XAE2 and GAE2 signals)

6256	
	ε value during automatic tool length measurement (M series) (for the XAE3 and GAE3 signals)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

These parameters set the relevant ε value during automatic tool compensation (T series) or automatic tool length measurement (M series).

NOTE
When the Reference axis (parameter No.1031) is Diameter specification, specify the diameter value. When the Reference axis (parameter No.1031) is Radius specification, specify the radius value.

Alarm and message

Number	Message	Description
PS0080	G37 MEASURING POSITION REACHED SIGNAL IS NOT PROPERLY INPUT	- For machining center series When the tool length measurement function (G37) is performed, a measuring position reached signal goes 1 in front of the area determined by the ε value specified in parameter No.6254. Alternatively, the signal does not go 1. - For lathe When the automatic tool compensation function (G36, G37) is used, a measuring position reached signals (XAE1, XAE2) does not go 1 within the range determined by the ε value specified in parameters Nos.6254 and 6255.
PS0081	G37 OFFSET NO. UNASSIGNED	- For machining center series The tool length measurement function (G37) is specified without specifying an H code. Correct the program. - For lathe The automatic tool compensation function (G36, G37) is specified without specifying an T code. Correct the program.
PS0082	G37 SPECIFIED WITH H CODE	- For machining center series The tool length measurement function (G37) is specified together with an H code in the same block. Correct the program. - For lathe The automatic tool compensation function (G36, G37) is specified together with an T code in the same block. Correct the program.
PS0083	G37 IMPROPER AXIS COMMAND	- For machining center series An error has been found in axis specification of the tool length measurement function (G37). Alternatively, a move command is specified as an incremental command. Correct the program. - For lathe An error has been found in axis specification of the automatic tool compensation function (G36, G37). Alternatively, a command is specified as an incremental command. Correct the program.

Note

NOTE

1. If an H code and G37 are specified in the same block, an alarm will be raised. Specify an H code before a block including G37.
2. A measurement speed (FP), γ, and ε are specified as parameters (FP: No.6241, γ: No.6251, ε: No.6254) by the machine tool builder. As ε, specify a positive value satisfying the condition of $\gamma > \varepsilon$.

NOTE

3 The compensation value is updated by the following formula:
New compensation value =(Current compensation value)+[(Current position of the tool along the specified axis when the measuring position reached signal is 1) - (specified measuring position)]
The following compensation values are updated:
(1) In an M series, the compensation value corresponding to the tool compensation number selected by an H code.
When offset memory A is used, the offset value is changed.
When offset memory B is used, the tool wear compensation value is changed.
When offset memory C is used, the tool wear compensation value for the H code is changed.
(2) In a T series, the compensation value corresponding to the tool compensation number selected by a T code and to the specified axis (X, Z) in G36, G37.

4 A delay or variation in detecting a measuring position reached signal is 0 to 2 ms just on the CNC side, excluding the PMC side. Accordingly, the measurement error is the sum of 2 ms and a delay or variation in transferring a measuring position reached signal on the PMC side (including receiver delay or variation), multiplied by the feedrate specified in parameter No. 6241.

5 A delay or variation after the detection of a measuring position reached signal until feed stop is 0 to 8 ms. When an overshoot is calculated, an acceleration/deceleration delay, servo delay, PMC delay must also be considered.

6 A measuring position reached signal is monitored just as a state not by the rising edge. Accordingly, if a measuring position reached signal is held to 1 and if the corresponding tool length measurement function (G37) is specified, the CNC will raise alarm PS0080 when a movement is made to a position at a distance of γ, in front of the measuring position.

7 When using this function, set the parameter EVO (bit 6 of parameter No. 5001) to zero. (If the tool compensation value of tool length compensation A or B is changed in the offset mode (G43 or G44), the new value becomes effective from a subsequent block including G43, G44, or an H code.)

Reference item

Manual name	Item name
DESCRIPTIONS (B-64692EN)	Automatic tool length measurement Automatic tool offset
OPERATOR'S MANUAL (Lathe system) (B-64694EN-1)	Automatic tool offset (G36,G37)
OPERATOR'S MANUAL (Machining center system) (B-64694EN-2)	Automatic tool length measurement (G37)

15.3 SKIP FUNCTION

15.3.1 Skip Function

Overview

Linear interpolation can be commanded by specifying axial move following the G31 command, like G01. If an external skip signal is input during the execution of this command, execution of the command is interrupted and the next block is executed.
The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.

The coordinates when the skip signal turned on are stored in the system variables (#100151 to #100182) for custom macros, so they can be used by custom macros. For a system with up to 20 axes, the system variables (#5061 to #5080) compatible with FS16 can also be used.

#100151 (#5061)	: First axis coordinate value
#100152 (#5062)	: Second axis coordinate value
:	
:	
#100170 (#5080)	: 20th axis coordinate value
#100171	: 21st axis coordinate value
#100172	: 22nd axis coordinate value
:	
:	
#100182	: 32nd axis coordinate value

- Feedrate

Two types of feedrate can be selected for the G31 block, using bit 1 (SFP) of parameter No.6207.

(1) If bit 1 (SFP) of parameter No.6207 is 0
Feedrate specified with address F (may be specified before or in the G31 block).

(2) If bit 1 (SFP) of parameter No.6207 is 1
Feedrate set for parameter No.6281.
Even if bit 7 (SKF) of parameter No.6200 is set to 1, dry run and automatic acceleration/deceleration are disabled when a feedrate with the G31 command is the feedrate which is set for parameter No.6821.

> ⚠ **CAUTION**
> To increase the precision of the tool position when the skip signal is input, feedrate override, dry run, and automatic acceleration/deceleration are disabled in moving by the skip function regardless of the feedrate which is specified as a feed per minute or a feed per rotation.
> To enable these functions, set bit 7 (SKF) of parameter No.6200 to 1.
> However, even if bit 7 (SKF) of parameter No.6200 is set to 1, dry run and automatic acceleration/deceleration are disabled when bit 1 (SFP) of parameter No.6207 is set to 1.

- Acc./Dec. after interpolation of the skip function

To enable the Acc./Dec. after cutting feed interpolation of the skip function, set bit 7 (SKF) of parameter No. 6200 to 1. When SKF is 0, the Acc./Dec. after cutting feed interpolation of the skip function is disabled.
If the Acc./Dec. type is set by bit 3 (ASL) of parameter No. 6210 and the time constant is set by bit 4 (ASB) of parameter No. 6210 when SKF is 1, it is possible to set a separate Acc./Dec. type and time constant that are different from those of the normal Acc./Dec. after cutting feed interpolation. When ASB is 0 and ASL is 0, the normal Acc./Dec. after cutting feed interpolation is assumed.
In this case, the Acc./Dec. type set by bit 0 (CTLx) of parameter No. 1610 and bit 1 (CTBx) of parameter No. 1610 and the time constant set by parameter No. 1622 are enabled.

- Skip signal

In the skip function, skip signal which is valid is as follows.

Table 15.3.1 (a) Whether the skip signals are enabled or disabled

Parameter	Bit 4 (IGX) of parameter No. 6201	Bit 0 (GSK) of parameter No. 6200	Bit 7 (SKPXE) of parameter No .6201	Skip signal SKIPP	Skip signal SKIP
Setting	0	0	0	Disabled	Enabled
	0	1	0	Enabled	Enabled
	0	0	1	Disabled	Disabled
	0	1	1	Enabled	Disabled
	1	0	0	Disabled	Disabled
	1	1	0	Disabled	Disabled
	1	0	1	Disabled	Disabled
	1	1	1	Disabled	Disabled

Bit 4 (IGX) of parameter No. 6201 is valid for the skip function using high-speed skip signals (when bit 4 (HSS) of parameter No. 6200 is set to 1) or for the multistage skip function using high-speed skip signals (when bit 5 (SLS) of parameter No. 6200 is set to 1).
And, when bit 1 (SK0) of parameter No. 6200 is set to 1, Skip signal is valid when the skip signal SKIP is 0.

- Accumulated pulses and positional deviation due to acceleration/deceleration

When bit 1 (SEB) of parameter No. 6201 is set to 1, the accumulated pulses and positional deviation due to acceleration/deceleration are considered and compensated.

- Skip block signal

Skip block signal SKPEXE<Fn537.3> becomes “1” while executing G31 block.

Signal

Skip Signals SKIP <X0004.7>, SKIPP <Gn006.6>

[Classification] Input signal

[Function] This signal terminates skip cutting. That is, the position where a skip signal turns to 1 in a block containing G31 is stored in a custom macro variable, and the move command of the block is terminated at the same time.

[Operation] When a skip signal turns to 1, the control unit works as described below.

- When a block contains a skip cutting command G31, the control unit reads and stores the current position of the specified axis at that time. The control unit stops the axis, then cancels the remaining distance that the block was supposed to be moved.

NOTE

1. The skip signal width requires at least 10 msec.
2. The delay or variation on the CNC side (excluding the PMC side) in detecting the skip signal is 0 to 2 msec. Therefore, the measurement error is the sum of 2 msec and the delay or variation (including the delay or variation of the receiver) of the skip signal on the PMC side, multiplied by the feedrate.
3. The delay or variation until feed stops after detection of the skip signal is 0 to 8 msec. To calculate the overshoot, further consider delay in acceleration/deceleration, delay in servo, delay on the PMC side.
4. The CNC directly reads the skip signal SKIP <X0004.7> from the machine tool, the PMC is no longer requires to process the signal.
5. If the skip function G31 is not used, the PMC can use the signal terminal SKIP <X0004.7> corresponding to the skip signal as a general purpose input signal.
6. When the skip signal is monitored, the state rather than the rising edge is sampled. Therefore, when the level remains 1, the skip condition is assumed to be met immediately after skip cutting is specified next.
7. Address <X0004>including skip signal SKIP <X0004.7> can be assigned to the X addresses that were set for parameters Nos. 3012 and 3019 when bit 2 (XSG) of parameter No. 3008 is 1.
8. When bit 2 (XSG) of parameter No. 3008 is 0, the address of the skip signal SKIP is <X0013.7> in case of the 2nd path and <X0011.7> in case of the 3rd path.
9. When the Multi-step skip function is effective, the bit 0 (1S1) of parameter No.6202 must be set to 1.

Skip Block Signal SKPEXE <Fn537.3>

[Classification] Output signal

[Function] This signal indicates that G31 block is executing.

[Operation] This signal becomes “1” while executing G31 block.
Other cases become “0”.

NOTE

When G31 block is stopped by the feed hold stop, this signal is “1”. Though this signal becomes “0” when the skip signal is input, this signal becomes “1” again when restarting from the feed hold stop.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X0004	SKIP							
X0011	SKIP#3							
X0013	SKIP#2							
Gn006		SKIPP						
Fn537					SKPEXE			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3008						XSG		

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#2 XSG A signal assigned to an X address is:
0: Fixed at the address.
1: Able to be reassigned to an arbitrary X address.

> **NOTE**
> When this parameter is set to 1, set parameters Nos. 3013, 3014, 3012, and 3019. If parameters Nos. 3013 and 3014 are not set, the deceleration signal for reference position return is assigned to bit 0 of <X0000>. If parameters Nos. 3012 and 3019 are not set, the skip signal, the PMC axis control skip signal, the measuring position reached signal, the interlock signal for each axis direction, and the tool compensation value write signal are assigned to <X0000>.

> ⚠ **WARNING**
> Please set an appropriate value to parameters (Nos. 3013, 3014, 3012, 3019) beforehand when you set 1 to this parameter.
> If an input signal of the set address in parameters (Nos. 3013, 3014, 3012, 3019) is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used X address.

3012	Skip signal assignment address

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 727
Set an X address to which the skip signal SKIPn is to be assigned.

> **NOTE**
> This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
> Depending on the configuration of the I/O Link, the actually usable X addresses are:
> <X0000 to X0127>, <X0200 to X0327>, <X0400 to X0527>, <X0600 to X0727>

> ⚠ **WARNING**
> If you used the set address in this parameter by another usage, it may cause an unexpected machine behavior.
> Be careful enough that you must prevent the competition of the used X address.

3019	Address to which the PMC axis control skip signal, measuring position reached signal, and tool offset write signals are assigned

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 727

Set an X address to which the PMC axis control skip signal ESKIP, measuring position reached signals (XAE, YAE, and ZAE (M series) or XAE and ZAE (T series)), and tool offset write signals (±MIT1 and ±MIT2 (T series)) are to be assigned.

Example 1. When parameter No. 3012 is set to 5 and parameter No. 3019 is set to 6

When bit 2 (XSG) of parameter No. 3008 is 1, the PMC axis control skip signal, and measuring position reached signal are allocated to X0006 and the skip signal is allocated to X0005.

	#7	#6	#5	#4	#3	#2	#1	#0	
X0005	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	(T series)
	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	(M series)

	#7	#6	#5	#4	#3	#2	#1	#0	
X0006		ESKIP					ZAE	XAE	(T series)
		ESKIP				ZAE	YAE	XAE	(M series)

Example 2. When parameter No. 3012 is set to 5 and parameter No. 3019 is set to 5

When bit 2 (XSG) of parameter No. 3008 is 1, the PMC axis control skip signal, measuring position reached signal, and skip signal are allocated to X0005.

	#7	#6	#5	#4	#3	#2	#1	#0	
X0005	SKIP	ESKIP SKIP6	 SKIP5	 SKIP4	 SKIP3	 SKIP2	ZAE SKIP8	XAE SKIP7	(T series)
	SKIP	ESKIP SKIP6	SKIP5	SKIP4	SKIP3	ZAE SKIP2	YAE SKIP8	XAE SKIP7	(M series)

> **NOTE**
> This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
> Depending on the configuration of the I/O Link, the actually usable X addresses are:
> <X0000 to X0127>, <X0200 to X0327>, <X0400 to X0527>, <X0600 to X0727>

> ⚠ **WARNING**
> If you used the set address in this parameter by another usage, it may cause an unexpected machine behavior.
> Be careful enough that you must prevent the competition of the used X address.

	#7	#6	#5	#4	#3	#2	#1	#0
6200	SKF			HSS			SK0	GSK

[Input type] Parameter input
[Data type] Bit path

#0 GSK As a skip signal, the skip signal SKIPP is:
0: Invalid.
1: Valid.

#1 SK0 This parameter specifies whether the skip signal is made valid under the state of the skip signal SKIP and the multistage skip signals SKIP2 to SKIP8.
0: Skip signal is valid when these signals are 1.
1: Skip signal is valid when these signals are 0.

#4 HSS 0: The skip function does not use high-speed skip signals while skip signals are input. (The conventional skip signal is used.)
1: The step skip function uses high-speed skip signals while skip signals are input.

#7 SKF Dry run, override, and automatic acceleration/deceleration for G31 skip command
0: Disabled
1: Enabled

> ⚠ **CAUTION**
> 1 When bit 7 (SKF) of parameter No.6200 is set to 0, feedrate override, dry run, and automatic acceleration/deceleration are disabled in moving by the skip function regardless of the feedrate which is specified as a feed per minute or a feed per rotation.
> 2 Even if bit 7 (SKF) of parameter No.6200 is set to 1, dry run and automatic acceleration/deceleration are disabled when bit 1 (SFP) of parameter No.6207 is set to 1. When the multi-step skip is used and bit 2 (SFN) of parameter No.6207 is set to 1, dry run and automatic acceleration/deceleration are similarly disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
6201	SKPXE			IGX			SEB	

[Input type] Parameter input
[Data type] Bit path

#1 SEB When a skip signal, high-speed skip signal or measuring position reached signal goes on while the skip function, the continuous high-speed skip function or the automatic tool length measurement (M series) or automatic tool compensation (T series) is used, the accumulated pulses and positional deviation due to acceleration/deceleration are:
0: Ignored.
1: Considered and compensated.
The accumulated pulses and positional deviation due to actual acceleration/deceleration when the skip signal, high-speed skip signal or measuring position reached signal goes on are considered to obtain the position at which the signal is input.

#4 IGX When the high-speed skip function is used, SKIP, SKIPP, and SKIP2 to SKIP8 are:
0: Enabled as skip signals.
1: Disabled as skip signals.

#7 SKPXE For the skip function (G31), the skip signal SKIP is:
0: Enabled.
1: Disabled.

Whether the skip signals are enabled or disabled

Parameter	Bit 4 (IGX) of parameter No. 6201	Bit 0 (GSK) of parameter No. 6200	Bit 7 (SKPXE) of parameter No .6201	Skip signal SKIPP	Skip signal SKIP	Multistage skip signals SKIP2-SKIP8
Setting	0	0	0	Disabled	Enabled	Enabled
	0	1	0	Enabled	Enabled	Enabled
	0	0	1	Disabled	Disabled	Enabled
	0	1	1	Enabled	Disabled	Enabled
	1	0	0	Disabled	Disabled	Disabled
	1	1	0	Disabled	Disabled	Disabled
	1	0	1	Disabled	Disabled	Disabled
	1	1	1	Disabled	Disabled	Disabled

Bit 4 (IGX) of parameter No. 6201 is valid for the skip function using high-speed skip signals (when bit 4 (HSS) of parameter No. 6200 is set to 1) or for the multistage skip function using high-speed skip signals (when bit 5 (SLS) of parameter No. 6200 is set to 1).
To use multistage skip signals, the multistage skip function is required.

	#7	#6	#5	#4	#3	#2	#1	#0
6202								1S1

[Input type] Parameter input
[Data type] Bit path

1S1 This parameter specify whether to enable or disable each high-speed skip signal when the G31 skip command is issued.
0: The high-speed skip signal HDI0 is disabled.
1: The high-speed skip signal HDI0 is enabled.

NOTE
1 Do not specify the same signal simultaneously for different paths.
2 At all of the following conditions, the bit 0 (1S1) of parameter No.6202 must be set to 1.
- The Multi-step skip function is effective.
- The High-speed skip signals are not used.
- Skip Signals SKIP<X0004.7>, SKIPP<Gn006.6> are used.

	#7	#6	#5	#4	#3	#2	#1	#0
6207							SFP	

[Input type] Parameter input
[Data type] Bit path

#1 SFP The feedrate used when the skip function (G31) is being executed is:
0: Feedrate of a programmed F code.
1: Feedrate set in parameter No. 6281.

> **NOTE**
> For the multi-stage skip function and high-speed skip, see the description of bit 2 (SFN) of parameter No. 6207.

	#7	#6	#5	#4	#3	#2	#1	#0
6210				ASB	ASL			

[Input type] Parameter input
[Data type] Bit path

#3 ASL
#4 ASB The ASB and ASL bits set the type and time constant of acceleration/deceleration after interpolation in the skip function as follows:

ASB	ASL	Type of acceleration/ deceleration	Parameter No. for time constant
0	1	Linear type	Parameter No. 6280
1	–	Bell-shaped	
0	0	This function is disabled(NOTE).	

When bell-shaped acceleration/deceleration is specified, T1=T/2 and T2=T/2 are obtained as with normal acceleration/deceleration after cutting feed interpolation, where T is the time constant. Therefore, the acceleration/deceleration type includes no linear part.

> **NOTE**
> In this case, the acceleration/deceleration type is set in bits 0 and 1 of parameter No. 1610, and the time constant is set in parameter No. 1622.

6280	Time constant for acceleration/deceleration after interpolation for the skip function for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 512

This parameter sets a time constant for acceleration/deceleration after interpolation for the skip function for each axis.
This parameter is valid when bit 3 (ASB) of parameter No. 6210 or bit 4 (ASL) of parameter No. 6210 is set to 1.

6281	Feedrate for the skip function (G31)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets a feedrate for the skip function (G31). This parameter is valid when bit 1 (SFP) of parameter No. 6207 is set to 1.

> **NOTE**
> For the multi-stage skip function and high-speed skip, see the description of parameters Nos. 6282 to 6285.

Alarm and message

Number	Message	Description
PS0035	CAN NOT COMMANDED G31	- G31 cannot be specified. This alarm is generated when a G code (such as for cutter or tool-nose radius compensation) of group 07 is not canceled. - A torque limit skip was not specified in a torque limit skip command (G31P98 or P99). Specify the torque limit skip in the PMC window or the like. Or, specify the torque limit override by address Q.

Note

> **NOTE**
> This function is disabled for the synchronous slave axis and enabled for axes not related to composite control.

15.3.2 Multiple Axis Command Skip Function

Overview

In a G31 block, the move command can be executed for multiple axes. When the skip signal is input externally during execution of this command, the execution is aborted for all axes and the next block is executed.

The skip positions of all of the specified axes are set in macro variables #5061 to #5080 (#100151 to #100182 for a system having 20 or more axes) corresponding to the axes.

Note

> **NOTE**
> This function is disabled for the synchronous slave axis and enabled for axes not related to composite control.

15.3.3 High-speed Skip

Overview

The skip function operates based on a high-speed skip signal (HDI0 to HDI7 : connected directly to the CNC; not via the PMC) instead of an ordinary skip signal. In this case, up to eight signals can be input. (Either can be enabled/disabled, using parameters HSS and IGX (bit 4 of parameter No. 6201 and bit 4 of parameter No. 6200).) If a high-speed skip signal is used, up to eight signals can be input.

Delay and error of skip signal input is 0 to 2 msec at the CNC side (not considering those at the PMC side).
This high-speed skip signal input function keeps this value to 0.1 msec or less, thus allowing high precision measurement.

Explanation

High–speed skip signal

The high-speed skip signal to be used is selected by bits 0 to 7 (1S1 to 1S8) of parameter No. 6202.
The high-speed skip signal is assumed to be input on the rising or falling edge. However, when the skip cutting is specified, the skip signal is monitored as a state. Therefore, skip operation is performed immediately when the skip cutting is specified with the high-speed skip signal input. The rising or falling edges of the high-speed skip signal can be selected as a trigger by setting bit 6 (SRE) of parameter No. 6200.

If setting bits 1 (SEB) of parameter No. 6201, the accumulated pulses and positional deviation due to acceleration/deceleration are considered and compensated.

Feedrate

Two types of feedrate can be selected for the high-speed skip block, using bit 2 (SFN) of parameter No.6207.

(1) If bit 2 (SFN) of parameter No.6207 is 0
Feedrate specified with address F (may be specified before or in the high-speed skip block).

(2) If bit 2 (SFN) of parameter No.6207 is 1
Feedrate set for parameters Nos.6282 to 6285.
Even if bit 7 (SKF) of parameter No.6200 is set to 1, dry run and automatic acceleration/deceleration are disabled when a feedrate with the G31 command is the feedrate which is set for parameters Nos.6282 to 6285.

> ⚠ **CAUTION**
> To increase the precision of the tool position when the high-speed skip signal is input, feedrate override, dry run, and automatic acceleration/deceleration are disabled in moving by the skip function regardless of the feedrate which is specified as a feed per minute or a feed per rotation.
> To enable these functions, set bit 7 (SKF) of parameter No.6200 to 1.
> However, even if bit 7 (SKF) of parameter No.6200 is set to 1, dry run and automatic acceleration/deceleration are disabled when bit 2 (SFN) of parameter No.6207 is set to 1.

Signal

High-Speed Skip Status Signals HDO0 to HDO7<Fn122>

[Classification] Output signal

[Function] This signal informs the PMC of the input status of the high-speed skip signal. The signal-to-bit correspondence is as follows:

High-speed skip signal	Bit name
HDI0	HDO0
HDI1	HDO1
HDI2	HDO2
HDI3	HDO3
HDI4	HDO4
HDI5	HDO5
HDI6	HDO6
HDI7	HDO7

[Output cond.] Each bit is set to 1 when:
- The corresponding high-speed skip signal is logical “1”.

Each bit is set to 0 when:
- The corresponding high-speed skip signal is logical “0”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn122	HDO7	HDO6	HDO5	HDO4	HDO3	HDO2	HDO1	HDO0

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200		SRE		HSS				

[Input type] Parameter input
[Data type] Bit path

#4 HSS 0: The skip function does not use high-speed skip signals while skip signals are input. (The conventional skip signal is used.)
1: The step skip function uses high-speed skip signals while skip signals are input.

#6 SRE When a high-speed skip signal is used:
0: The signal is assumed to be input on the rising edge (contact open → close).
1: The signal is assumed to be input on the falling edge (contact close → open).

	#7	#6	#5	#4	#3	#2	#1	#0
6201							SEB	

[Input type] Parameter input
[Data type] Bit path

#1 SEB When a skip signal, high-speed skip signal or measuring position reached signal goes on while the skip function, the continuous high-speed skip function or the automatic tool length measurement (M series) or automatic tool compensation (T series) is used, the accumulated pulses and positional deviation due to acceleration/deceleration are:
0: Ignored.
1: Considered and compensated.

The accumulated pulses and positional deviation due to actual acceleration/deceleration when the skip signal, high-speed skip signal or measuring position reached signal goes on are considered to obtain the position at which the signal is input.

	#7	#6	#5	#4	#3	#2	#1	#0
6202	1S8	1S7	1S6	1S5	1S4	1S3	1S2	1S1

[Input type] Parameter input
[Data type] Bit path

1S1 to 1S8 These parameters specify whether to enable or disable each high-speed skip signal when the G31 skip command is issued.
The following table shows the correspondence between the bits, input signals, and commands.
The settings of the bits have the following meaning :
0: The high-speed skip signal corresponding to a bit is disabled.
1: The high-speed skip signal corresponding to a bit is enabled.

Parameter	High-speed skip signals	Parameter	High-speed skip signals
1S1	HDI0	1S5	HDI4
1S2	HDI1	1S6	HDI5
1S3	HDI2	1S7	HDI6
1S4	HDI3	1S8	HDI7

NOTE
1 Do not specify the same signal simultaneously for different paths.
2 At all of the following conditions, the bit 0 (1S1) of parameter No.6202 must be set to 1.
- The Multi-step skip function is effective.
- The High-speed skip signals are not used.
- Skip Signals SKIP<X0004.7>, SKIPP<Gn006.6> are used.

	#7	#6	#5	#4	#3	#2	#1	#0
6207						SFN		

[Input type] Parameter input
[Data type] Bit path

#2 SFN The feedrate used when the skip function based on high-speed skip signals (with bit 4 (HSS) of parameter No. 6200 set to 1) or the multi-skip function is being executed is:
0: Feedrate of a programmed F code.
1: Feedrate set in a parameter from parameters Nos. 6282 to 6285.

NOTE
For not the multistage skip function, but the skip function using no high-speed skip signals (when bit 4 (HSS) of parameter No. 6200 is set to 0), see the description of bit 1 (SFP) of parameter No. 6207.

6282	Feedrate for the skip function (G31, G31 P1)
6283	Feedrate for the skip function (G31 P2)
6284	Feedrate for the skip function (G31 P3)
6285	Feedrate for the skip function (G31 P4)

[Input type] Parameter input
[Data type] Real path

[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Each of these parameters sets a feedrate for each skip function G code. These parameters are valid when bit 2 (SFN) of parameter No. 6207 is set to 1.

Alarm and message

Number	Message	Description
PS0373	ILLEGAL HIGH-SPEED SKIP SIGNAL SELECTED	In the skip commands (G31, G31P1 to G31P4) and dwell commands (G04, G04Q1 to G04Q4), the same high-speed signal is selected in different paths.

Note

NOTE
1 This function is disabled for the synchronous slave axis and enabled for axes not related to composite control.
2 Do not specify the same high-speed skip signal simultaneously for different paths. In the skip commands (G31, G31P1 to G31P4) and dwell commands (G04, G04Q1 to G04Q4), if the same high-speed signal is selected in different paths, an alarm (PS0373) "ILLEGAL HIGH-SPEED SKIP SIGNAL SELECTED" is issued.

Notes on using high speed skip

- Overview

Note that the contact state during a skip is different from that of Series 0*i*-C in the high speed skip signal of Series 0*i*-F Plus.

- Series 0*i*-F Plus :
  The skip signal is assumed to be input when an open contact is closed.
- Series 0*i*-C :
  The skip signal is assumed to be input when an closed contact is opened.

- Detail

The difference in the specification of the high speed skip signal between Series 0*i*-F Plus and Series 0*i*-C is as follows.

- For Series 0*i*-F Plus
  The high-speed skip signal is assumed to be 1 when the input voltage is at a low level and assumed to be 0 at a high level. That is, the skip signal is assumed to be 1 when a contact is closed.
  Therefore the skip signal is input when an open contact is closed.
- For Series 0*i*-C
  The high-speed skip signal is assumed to be 0 when the input voltage is at a low level and assumed to be 1 at a high level. That is, the skip signal is assumed to be 1 when a contact is opened.
  Therefore the skip signal is input when a closed contact is opened.
  When the wiring designed for 0*i*-C is applied to 0*i*-F Plus, the modification of wiring is not necessary by setting the following parameter.
  Depending on the detector to be used, the settings on the detector are used instead of parameter settings to switch between contact A and contact B of for output signals. For details, see the manual of the detector.

- Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200		SRE						

[Input type] Parameter input
[Data type] Bit path

#6 SRE When a high-speed skip signal is used:
0: The signal is assumed to be input on the rising edge (contact open → close).
1: The signal is assumed to be input on the falling edge (contact close → open).

- Signal

High-speed skip status signal HDO0 to HDO7<Fn122>

	#7	#6	#5	#4	#3	#2	#1	#0
Fn122	HDO7	HDO6	HDO5	HDO4	HDO3	HDO2	HDO1	HDO0

[Classification] Output signal
[Function] This signal informs the PMC of the input status of the high-speed skip signal.
[Output cond.] 1: When the contact of the high-speed skip signal is closed.
(The input voltage is at a low level.)
0: When the contact of the high-speed skip signal is open.
(The input voltage is at a high level.)

15.3.4 Continuous High-Speed Skip Function

Overview

The continuous high-speed skip function enables reading of absolute coordinates by using the high-speed skip signals (HDI0 to HDI7). Once a high-speed skip signal has been input in a G31P90 block, absolute coordinates are written to custom macro variables #5061 to #5080. For a system with more than 20 axes, #100151 to #100182 are used. The input of a skip signal does not stop axial movement, thus continuous reading of the coordinates of two or more points is possible.
Both the rising and falling edges of the high-speed skip signal can be used as a trigger by setting bit 5 (CSE) of parameter No. 6201.

Explanation

Custom macro variables

Once a high–speed skip signal has been input in a G31P90 block, absolute coordinates are written to custom macro variables #5061 to #5080. For a system with more than 20 axes, #100151 to #100182 are used.

These variables are immediately updated once the tool reaches the next skip position. The feedrate must, therefore, be specified so that the tool does not reach the next skip position before the application completes reading of the variables.

System with up to 20 axes		System with more than 20 axes	
#5061	Coordinate along the first axis	#100151	Coordinate along the first axis
#5062	Coordinate along the second axis	#100152	Coordinate along the second axis
#5063	Coordinate along the third axis	#110153	Coordinate along the third axis
:		:	
#5079	Coordinate along the 19th axis	#100181	Coordinate along the 31st axis
#5080	Coordinate along the 20th axis	#100182	Coordinate along the 32nd axis

High–speed skip

This function is enabled only for high-speed skip signals.
The high-speed skip signal to be used is selected by bits 0 to 7 (9S1 to 9S8) of parameter No. 6208.

End of block

The G31P90 block is terminated when the axis reaches the end point.

Limitations

Only one axis can be specified in the block for the continuous high-speed skip function (G31P90). If two or more axes are specified, alarm PS5068, "FORMAT ERROR IN G31P90" is issued.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200		SRE		HSS				

[Input type] Parameter input
[Data type] Bit path

#4 HSS 0: The skip function does not use high-speed skip signals while skip signals are input. (The conventional skip signal is used.)
1: The step skip function uses high-speed skip signals while skip signals are input.

#6 SRE When a high-speed skip signal is used:
0: The signal is assumed to be input on the rising edge (contact open → close).
1: The signal is assumed to be input on the falling edge (contact close → open).

	#7	#6	#5	#4	#3	#2	#1	#0
6201			CSE				SEB	

[Input type] Parameter input
[Data type] Bit path

#1 SEB When a skip signal, high-speed skip signal or measuring position reached signal goes on while the skip function, the continuous high-speed skip function or the automatic tool length measurement (M series) or automatic tool compensation (T series) is used, the accumulated pulses and positional deviation due to acceleration/deceleration are:
0: Ignored.
1: Considered and compensated.
The accumulated pulses and positional deviation due to actual acceleration/deceleration when the skip signal, high-speed skip signal or measuring position reached signal goes on are considered to obtain the position at which the signal is input.

#5 CSE For the continuous high-speed skip command, high-speed skip signals are:
0: Effective at either a rising or falling edge (depending on the setting of bit 6 (SRE) of parameter No. 6200).
1: Effective at both the rising and falling edges.

	#7	#6	#5	#4	#3	#2	#1	#0
6208	**9S8**	**9S7**	**9S6**	**9S5**	**9S4**	**9S3**	**9S2**	**9S1**

[Input type] Parameter input
[Data type] Bit path

9S1 to 9S8 Specify which high-speed skip signal is enabled for the continuous high-speed skip command G31P90 or the EGB skip command G31.8.
The settings of each bit have the following meaning:
0: The high-speed skip signal corresponding to the bit is disabled.
1: The high-speed skip signal corresponding to the bit is enabled.
The bits correspond to signals as follows:

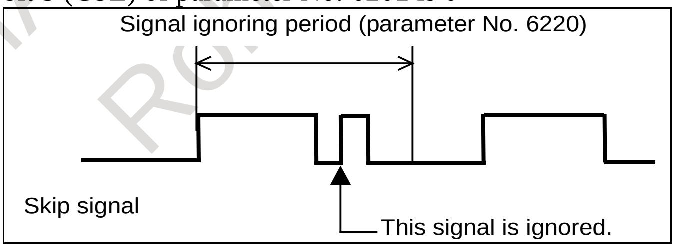
Parameter	High-speed skip signal	Parameter	High-speed skip signal
9S1	HDI0	9S5	HDI4
9S2	HDI1	9S6	HDI5
9S3	HDI2	9S7	HDI6
9S4	HDI3	9S8	HDI7

6220	Period during which skip signal input is ignored for the continuous high-speed skip function and EGB axis skip function

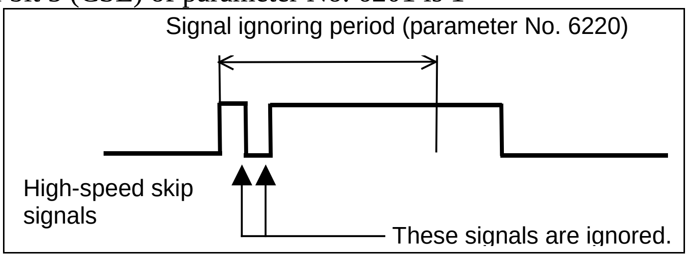
[Input type] Parameter input
[Data type] Byte path
[Unit of data] 8msec
[Valid data range] 3 to 127(× 8msec)

This parameter specifies the period from when a skip signal is input to when the next skip signal can be input for the continuous high-speed skip function and EGB axis skip function. This parameter is used to ignore chattering in skip signals.
If a value that falls outside the valid range is specified, the setting is assumed to be 24 msec.

Example 1)
When bit 5 (CSE) of parameter No. 6201 is 0



Example 2)
When bit 5 (CSE) of parameter No. 6201 is 1



Signal

In a continuous high-speed skip, the high-speed skip signal is used. For details on the signal, see Section, "High-Speed Skip Signal".

Alarm and message

Number	Message	Description
PS5068	FORMAT ERROR IN G31P90	No travel axis was specified. Two or more travel axes were specified.
PS0369	G31 FORMAT ERROR	- No axis is specified or tow or more axes are specified in the torque limit switch instruction (G31P98/P99). - The specified torque Q value in the torque limit switch instruction is out of range. The torque Q range is 1 to 254. - The high-speed continuous skip is not effective.

Note

NOTE
To use this function, the continuous high-speed skip, high-speed skip and custom macro are required.

15.3.5 Multi-step Skip

Overview

The multi-step skip function stores the coordinate values when skip signals are input in the block where G31P1 to G31P4 are specified, and skips the remaining movement. In the block where Q1 to Q4 are specified after G04, dwell can be skipped when skip signals are input.
A skip signal from equipment such as a fixed-dimension size measuring instrument can be used to skip programs being executed.
In plunge grinding, for example, a series of operations from rough machining to spark-out can be performed automatically by applying a skip signal each time rough machining, semi-fine machining, fine-machining, or spark-out operation is completed.

When the multi-step skip function is effective, to use high-speed skip signals, set bit 5 (SLS) of parameter No.6200 to 1.

- Feedrate

Two types of feedrate can be selected for the multi-step skip block, using bit 2 (SFN) of parameter No.6207.

(1) If bit 2 (SFN) of parameter No.6207 is 0
Feedrate specified with address F (may be specified before or in the multi-step skip block).

(2) If bit 2 (SFN) of parameter No.6207 is 1
Feedrate set for parameters Nos.6282 to 6285.
Even if bit 7 (SKF) of parameter No.6200 is set to 1, dry run and automatic acceleration/deceleration are disabled when a feedrate with the G31 command is the feedrate which is set for parameters Nos.6282 to 6285.

⚠ CAUTION
To increase the precision of the tool position when the skip signal is input, feedrate override, dry run, and automatic acceleration/deceleration are disabled in moving by the skip function regardless of the feedrate which is specified as a feed per minute or a feed per rotation.
To enable these functions, set bit 7 (SKF) of parameter No.6200 to 1.
However, even if bit 7 (SKF) of parameter No.6200 is set to 1, dry run and automatic acceleration/deceleration are disabled when bit 2 (SFN) of parameter No.6207 is set to 1.

Signal

Skip signals SKIPP<Gn006.6>,SKIP<X0004.7>,SKIP2<X0004.2>,SKIP3<X0004.3> SKIP4<X0004.4>,SKIP5<X0004.5>,SKIP6<X0004.6>,SKIP7<X0004.0> SKIP8<X0004.1>

[Classification] Input signal

[Function] These signals terminate skip cutting. That is, the position where a skip signal turns to 1 in a command program block containing G31P1 (or G31), G31P2, or G31P3, G31P4 is stored in a custom macro variable, and the move command of the block is terminated at the same time. Furthermore, in a block containing G04, G04Q1, G04Q2, G04Q3 or G04Q4, the dwell command of the block is terminated.

In either case, until all other commands (such as miscellaneous functions) of the block are completed, machining never proceeds to the next block.

Parameters Nos. 6202 to 6206 are used to select the signals to be enabled from eight signals. The eight skip signals can correspond to the G codes on a one-to-one basis. One skip signal can also be made applicable to multiple G codes. Conversely, multiple skip signals can be made applicable to one G code.

[Operation] When a skip signal turns to 1, the control unit functions as described below.

- When a block contains a G code from (G31, G31P1 to P4) for skip cutting, and the skip signal is made applicable by parameter setting to the command, the control unit reads and stores the current position of the specified axis at that time. The control unit stops the axis, then cancels the remaining distance that the axis was supposed to be moved in that block.
- When a block contains a G04, or G04Q1 to Q4 code for dwell, and the skip signal is made applicable by parameter setting to the command, the control unit stops dwell operation, and cancels any remaining dwell time.

NOTE

1. The skip signal width requires at least 10 msec.
2. The delay or variation on the CNC side (excluding the PMC side) in detecting the skip signal is 0 to 2 msec. Therefore, the measurement error is the sum of 2 msec and the delay or variation (including the delay or variation of the receiver) of the skip signal on the PMC side, multiplied by the feedrate.
3. The delay or variation until feed stops after detection of the skip signal is 0 to 8 msec. To calculate the overshoot, further consider delay in acceleration/deceleration, delay in servo, delay on the PMC side.
4. The G codes for skip cutting (G31P1, G31P2, G31P3, and G31P4) are the same except for their correspondence with skip signals. G31 and G31P1 are equivalent.
5. The skip signal is not monitored for a rising edge, but for its state. So, if a skip signal continues to be 1, a skip condition is assumed to be satisfied immediately when the next skip cutting or dwell operation is specified.
6. Address <X0004> can be allocated optionally to the X addresses set in parameter No. 3012 and parameter No. 3019 when bit 2 (XSG) of parameter No. 3008 is 1.
7. In case that the command value of G31P or G04Q is incorrect, or in the absence of the Multi-step skip function, alarm PS0370 “G31P/G04Q ERROR” occurs.

> **NOTE**
> 8 The same high-speed signal cannot be selected in different paths. In that case, alarm PS0373 “ILLEGAL HIGH-SPEED SKIP SIGNAL SELECTED” occurs.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
Gn006		SKIPP							

	#7	#6	#5	#4	#3	#2	#1	#0	
X0004	SKIP	ESKIP / SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	ZAE / SKIP8	XAE / SKIP7	(T series)
	SKIP	ESKIP / SKIP6	SKIP5	SKIP4	SKIP3	ZAE / SKIP2	YAE / SKIP8	XAE / SKIP7	(M series)

	#7	#6	#5	#4	#3	#2	#1	#0	
X0011	SKIP[#3]	ESKIP[#3] / SKIP6[#3]	SKIP5[#3]	SKIP4[#3]	SKIP3[#3]	SKIP2[#3]	ZAE[#3] / SKIP8[#3]	XAE[#3] / SKIP7[#3]	(T series)
	SKIP[#3]	ESKIP[#3] / SKIP6[#3]	SKIP5[#3]	SKIP4[#3]	SKIP3[#3]	ZAE[#3] / SKIP2[#3]	YAE[#3] / SKIP8[#3]	XAE[#3] / SKIP7[#3]	(M series)

	#7	#6	#5	#4	#3	#2	#1	#0	
X0013	SKIP[#2]	ESKIP[#2] / SKIP6[#2]	SKIP5[#2]	SKIP4[#2]	SKIP3[#2]	SKIP2[#2]	ZAE[#2] / SKIP8[#2]	XAE[#2] / SKIP7[#2]	(T series)
	SKIP[#2]	ESKIP[#2] / SKIP6[#2]	SKIP5[#2]	SKIP4[#2]	SKIP3[#2]	ZAE[#2] / SKIP2[#2]	YAE[#2] / SKIP8[#2]	XAE[#2] / SKIP7[#2]	(M series)

> ⚠ **WARNING**
> 1 SKIP6 to SKIP8 are at the same addresses as skip signal ESKIP (PMC axis control) and measuring position reached signal XAE and ZAE (tool length automatic measurement). Be careful when using both. (T series)
> 2 SKIP2 and SKIP6 to SKIP8 are at the same addresses as skip signal ESKIP (PMC axis control) and measuring position reached signal XAE, YAE, and ZAE (tool length automatic measurement). Be careful when using both. (M series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3008						XSG		

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#2 XSG A signal assigned to an X address is:
0: Fixed at the address.
1: Able to be reassigned to an arbitrary X address.

NOTE

When this parameter is set to 1, set parameters Nos. 3013, 3014, 3012, and 3019. If parameters Nos. 3013 and 3014 are not set, the deceleration signal for reference position return is assigned to bit 0 of <X0000>. If parameters Nos. 3012 and 3019 are not set, the skip signal, the PMC axis control skip signal, the measuring position reached signal, the interlock signal for each axis direction, and the tool compensation value write signal are assigned to <X0000>.

⚠ **WARNING**

Please set an appropriate value to parameters (Nos. 3013, 3014, 3012, 3019) beforehand when you set 1 to this parameter.
If an input signal of the set address in parameters (Nos. 3013, 3014, 3012, 3019) is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used X address.

3012	Skip signal assignment address

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 727

Set an X address to which the skip signal SKIPn is to be assigned.

NOTE

This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
Depending on the configuration of the I/O Link, the actually usable X addresses are:
<X0000 to X0127>, <X0200 to X0327>, <X0400 to X0527>, <X0600 to X0727>

⚠ **WARNING**

If you used the set address in this parameter by another usage, it may cause an unexpected machine behavior.
Be careful enough that you must prevent the competition of the used X address.

3019	Address to which the PMC axis control skip signal, measuring position reached signal, and tool offset write signals are assigned

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 727

Set an X address to which the PMC axis control skip signal ESKIP, measuring position reached signals (XAE, YAE, and ZAE (M series) or XAE and ZAE (T series)), and tool offset write signals (±MIT1 and ±MIT2 (T series)) are to be assigned.

NOTE
This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
Depending on the configuration of the I/O Link, the actually usable X addresses are:
<X0000 to X0127>, <X0200 to X0327>, <X0400 to X0527>, <X0600 to X0727>

⚠ **WARNING**
If you used the set address in this parameter by another usage, it may cause an unexpected machine behavior.
Be careful enough that you must prevent the competition of the used X address.

	#7	#6	#5	#4	#3	#2	#1	#0
6200	SKF		SLS				SK0	GSK

[Input type] Parameter input
[Data type] Bit path

#0 GSK As a skip signal, the skip signal SKIPP is:
0: Invalid.
1: Valid.

#1 SK0 This parameter specifies whether the skip signal is made valid under the state of the skip signal SKIP and the multistage skip signals SKIP2 to SKIP8.
0: Skip signal is valid when these signals are “1”.
1: Skip signal is valid when these signals are “0”.

#5 SLS 0: The multi-step skip function does not use high-speed skip signals while skip signals are input. (The conventional skip signal is used.)
1: The multi-step skip function uses high-speed skip signals while skip signals are input.

> **NOTE**
> The skip signals (SKIP and SKIP2 to SKIP8) are valid regardless of the setting of this parameter. They can also be disabled using bit 4 (IGX) of parameter No. 6201.
> If you want to use high-speed skip signals when the multi-step skip function is used, set this parameter to 1.

#7 SKF Dry run, override, and automatic acceleration/deceleration for G31 skip command
0: Disabled
1: Enabled

> ⚠ **CAUTION**
> 1 When bit 7 (SKF) of parameter No.6200 is set to 0, feedrate override, dry run, and automatic acceleration/deceleration are disabled in moving by the skip function regardless of the feedrate which is specified as a feed per minute or a feed per rotation.
> 2 Even if bit 7 (SKF) of parameter No.6200 is set to 1, dry run and automatic acceleration/deceleration are disabled when bit 1 (SFP) of parameter No.6207 is set to 1. When the multi-step skip is used and bit 2 (SFN) of parameter No.6207 is set to 1, dry run and automatic acceleration/deceleration are similarly disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
6201	SKPXE			IGX			SEB	

[Input type] Parameter input
[Data type] Bit path

#1 SEB When a skip signal, high-speed skip signal or measuring position reached signal goes on while the skip function, the continuous high-speed skip function or the automatic tool length measurement (M series) or automatic tool compensation (T series) is used, the accumulated pulses and positional deviation due to acceleration/deceleration are:
0: Ignored.
1: Considered and compensated.
The accumulated pulses and positional deviation due to actual acceleration/deceleration when the skip signal, high-speed skip signal or measuring position reached signal goes on are considered to obtain the position at which the signal is input.

#4 IGX When the high-speed skip function is used, SKIP, SKIPP, and SKIP2 to SKIP8 are:
0: Enabled as skip signals.
1: Disabled as skip signals.

#7 SKPXE For the skip function (G31), the skip signal SKIP is:
0: Enabled.
1: Disabled.

Whether the skip signals are enabled or disabled

Parameter	Bit 4 (IGX) of parameter No. 6201	Bit 0 (GSK) of parameter No. 6200	Bit 7 (SKPXE) of parameter No .6201	Skip signal SKIPP	Skip signal SKIP	Multistage skip signals SKIP2-SKIP8
Setting	0	0	0	Disabled	Enabled	Enabled
	0	1	0	Enabled	Enabled	Enabled
	0	0	1	Disabled	Disabled	Enabled
	0	1	1	Enabled	Disabled	Enabled
	1	0	0	Disabled	Disabled	Disabled
	1	1	0	Disabled	Disabled	Disabled
	1	0	1	Disabled	Disabled	Disabled
	1	1	1	Disabled	Disabled	Disabled

Bit 4 (IGX) of parameter No. 6201 is valid for the skip function using high-speed skip signals (when bit 4 (HSS) of parameter No. 6200 is set to 1) or for the multistage skip function using high-speed skip signals (when bit 5 (SLS) of parameter No. 6200 is set to 1).
To use multistage skip signals, the multistage skip function is required.

	#7	#6	#5	#4	#3	#2	#1	#0
6202	1S8	1S7	1S6	1S5	1S4	1S3	1S2	1S1
6203	2S8	2S7	2S6	2S5	2S4	2S3	2S2	2S1
6204	3S8	3S7	3S6	3S5	3S4	3S3	3S2	3S1
6205	4S8	4S7	4S6	4S5	4S4	4S3	4S2	4S1
6206	DS8	DS7	DS6	DS5	DS4	DS3	DS2	DS1

[Input type] Parameter input
[Data type] Bit path

1S1to1S8, 2S1to2S8, 3S1to3S8, 4S1to4S8, DS1toDS8

Specify which skip signal is enabled when the skip command (G31, or G31P1 to G31P4) and the dwell command (G04, G04Q1 to G04Q4) are issued with the multi-step skip function.
The following table shows the correspondence between the bits, input signals, and commands.
The setting of the bits have the following meaning :
0: The skip signal corresponding to a bit is invalid.
1: The skip signal corresponding to a bit is enabled.

Multi-step skip function

Command / Input signal	G31 G31P1 G04Q1	G31P2 G04Q2	G31P3 G04Q3	G31P4 G04Q4	G04
SKIP/HDI0	1S1	2S1	3S1	4S1	DS1
SKIP2/HDI1	1S2	2S2	3S2	4S2	DS2
SKIP3/HDI2	1S3	2S3	3S3	4S3	DS3
SKIP4/HDI3	1S4	2S4	3S4	4S4	DS4
SKIP5/HDI4	1S5	2S5	3S5	4S5	DS5
SKIP6/HDI5	1S6	2S6	3S6	4S6	DS6
SKIP7/HDI6	1S7	2S7	3S7	4S7	DS7
SKIP8/HDI7	1S8	2S8	3S8	4S8	DS8

> **NOTE**
> 1 HDI0 to HDI7 are high-speed skip signals. Do not specify the same signal simultaneously for different paths.
> 2 At all of the following conditions, the bit 0 (1S1) of parameter No.6202 must be set to 1.
> - The Multi-step skip function is effective.
> - The High-speed skip signals are not used.
> - Skip Signals SKIP<X0004.7>, SKIPP<Gn006.6> are used.

When bit 0 (GSK) of parameter No. 6200 is set to 1, commands to be skipped can be selected by setting the following parameter:

Commands skipped by SKIPP signal <G006.6>

Parameter	Command skipped
When bit 0 (1S1) of parameter No. 6202 is set to 1	G31P1,G04Q1
When bit 0 (2S1) of parameter No. 6203 is set to 1	G31P2,G04Q2
When bit 0 (3S1) of parameter No. 6204 is set to 1	G31P3,G04Q3
When bit 0 (4S1) of parameter No. 6205 is set to 1	G31P4,G04Q4
When bit 0 (DS1) of parameter No. 6206 is set to 1	G04,G04Q1,G04Q2,G04Q3,G04Q4

	#7	#6	#5	#4	#3	#2	#1	#0
6207						SFN		

[Input type] Parameter input
[Data type] Bit path

#2 SFN The feedrate used when the skip function based on high-speed skip signals (with bit 4 (HSS) of parameter No. 6200 set to 1) or the multi-skip function is being executed is:
0: Feedrate of a programmed F code.
1: Feedrate set in a parameter from parameters Nos. 6282 to 6285.

> **NOTE**
> For not the multistage skip function, but the skip function using no high-speed skip signals (when bit 4 (HSS) of parameter No. 6200 is set to 0), see the description of bit 1 (SFP) of parameter No. 6207.

6282	**Feedrate for the skip function (G31, G31 P1)**
6283	**Feedrate for the skip function (G31 P2)**
6284	**Feedrate for the skip function (G31 P3)**
6285	**Feedrate for the skip function (G31 P4)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Each of these parameters sets a feedrate for each skip function G code. These parameters are valid when bit 2 (SFN) of parameter No. 6207 is set to 1.

Alarm and message

Number	Message	Description
PS0035	CAN NOT COMMANDED G31	- G31 cannot be specified. This alarm is generated when a G code (such as for cutter or tool-nose radius compensation) of group 07 is not canceled. - A torque limit skip was not specified in a torque limit skip command (G31P98 or P99). Specify the torque limit skip in the PMC window or the like. Or, specify the torque limit override by address Q.
PS0370	G31P/G04Q ERROR	1) The specified address P value for G31 is out of range. The address P range is 1 to 4 in a multistage skip function. 2) The specified address Q value for G04 is out of range. The address Q range is 1 to 4 in a multistage skip function. 3) P1-4 for G31, or Q1-4 for G04 was commanded without a multistage skip function. 4) <T series > The specified value of address P of G72 or G74 falls outside the range. Address P ranges from 1 to 4 in the multistage skip function. P1-4 was specified in G72 or G74 even though the multistage skip function is not present.
PS0373	ILLEGAL HIGH-SPEED SKIP SIGNAL SELECTED	In the skip commands (G31, G31P1 to G31P4) and dwell commands (G04, G04Q1 to G04Q4), the same high-speed signal is selected in different paths.

Note

NOTE

This function is disabled for the synchronous slave axis and enabled for axes not related to composite control.

15.3.6 Torque Limit Skip Function

Overview

When the movement command following G31 P99 (or G31 P98) is executed with the servo motor torque limit[*1] overridden[*2], cutting feed similar to linear interpolation (G01) can be performed. When the servo motor torque[*3] reaches the torque limit (overridden servo motor torque limit) by pushing or the skip signal (including the high-speed skip signal) is input during the movement, the remaining movement commands are canceled and then the next block is executed. (The operation that executes the next block by canceling the remaining movement command is called skip operation later.)

The servo motor torque limit can be overridden by:

(1) Executing the torque limit override command for the PMC window.

(2) Executing the address Q command in the same block in which G31 P99 (or G31 P98) exists.

*1: The servo motor torque limit (parameter No.2060) is automatically set according to the settings of the motor model. The override is applied to the maximum torque of the servo motor.

*2: Set the torque limit override by the window function (function code 152) of PMC. Refer to PMC PROGRAMMING MANUAL (B-64513EN) for window Functions.

*3: Servo motor torque can be traced by waveform diagnosis display. Refer to Series 0*i*-F Plus MAINTENANCE MANUAL (B-64695EN) for waveform diagnosis display.

In order to override the torque limit on the servo motor, set bit 0 (TQO) of parameter No. 6286 to 1.

If the bit 0 (TQI) of parameter No.1803 is set to 1, in-position check is disabled during torque limit.

If the bit 1 (TQA) of parameter No.1803 is set to 1, an excessive stop-time/move-time error is not checked during torque limit.

If setting parameter No.6221, torque limit reached signals are ignored for a set period of time.

Signal

Torque limit reached signals TRQL1 to TRQL 8<Fn114>

[Classification] Input signal

[Function] Indicates that the torque limit has been reached.

[Output cond.] Set to 1 when:

- The torque limit has been reached.

Set to 0 when:

- The torque limit has not been reached.

Indicates the signal for each axis. The value at the end of a signal name indicates the number of each control axis.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn114	TRQL8	TRQL7	TRQL6	TRQL5	TRQL4	TRQL3	TRQL2	TRQL1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1803							TQA	TQI

[Input type] Parameter input

[Data type] Bit path

#0 TQI Within a torque limit, an in-position check is:

0: Made.

1: Not made.

#1 TQA Within a torque limit, an excessive stop-time/move-time error is:

0: Checked.

1: Not checked.

	#7	#6	#5	#4	#3	#2	#1	#0
3008						XSG		

[Input type] Parameter input

[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#2 XSG A signal assigned to an X address is:

0: Fixed at the address.

1: Able to be reassigned to an arbitrary X address.

> **NOTE**
> When this parameter is set to 1, set parameters Nos. 3013, 3014, 3012, and 3019. If parameters Nos. 3013 and 3014 are not set, the deceleration signal for reference position return is assigned to bit 0 of <X0000>. If parameters Nos. 3012 and 3019 are not set, the skip signal, the PMC axis control skip signal, the measuring position reached signal, the interlock signal for each axis direction, and the tool compensation value write signal are assigned to <X0000>.

> ⚠ **WARNING**
> Please set an appropriate value to parameters (Nos. 3013, 3014, 3012, 3019) beforehand when you set 1 to this parameter.
> If an input signal of the set address in parameters (Nos. 3013, 3014, 3012, 3019) is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used X address.

3012	Skip signal assignment address

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 727

Set an X address to which the skip signal SKIPn is to be assigned.

> **NOTE**
> This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
> Depending on the configuration of the I/O Link, the actually usable X addresses are:
> <X0000 to X0127>, <X0200 to X0327>, <X0400 to X0527>, <X0600 to X0727>

> ⚠ **WARNING**
> If you used the set address in this parameter by another usage, it may cause an unexpected machine behavior.
> Be careful enough that you must prevent the competition of the used X address.

3019	Address to which the PMC axis control skip signal and measuring position reached signals are assigned

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 727

Set an X address to which the PMC axis control skip signal ESKIP and measuring position reached signals (XAE, YAE, and ZAE (M series) or XAE and ZAE (T series)) are to be assigned.

> **NOTE**
> This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
> Depending on the configuration of the I/O Link, the actually usable X addresses are:
> <X0000 to X0127>, <X0200 to X0327>, <X0400 to X0527>, <X0600 to X0727>

> ⚠ **WARNING**
> If you used the set address in this parameter by another usage, it may cause an unexpected machine behavior.
> Be careful enough that you must prevent the competition of the used X address.

	#7	#6	#5	#4	#3	#2	#1	#0
6200	SKF			HSS			SK0	GSK

[Input type] Parameter input
[Data type] Bit path

#0 GSK As a skip signal, the skip signal SKIPP is:
- 0: Invalid.
- 1: Valid.

#1 SK0 This parameter specifies whether the skip signal is made valid under the state of the skip signal SKIP and the multistage skip signals SKIP2 to SKIP8.
- 0: Skip signal is valid when these signals are “1”.
- 1: Skip signal is valid when these signals are “0”.

#4 HSS
- 0: The skip function does not use high-speed skip signals while skip signals are input. (The conventional skip signal is used.)
- 1: The step skip function uses high-speed skip signals while skip signals are input.

#7 SKF Dry run, override, and automatic acceleration/deceleration for G31 skip command
- 0: Disabled
- 1: Enabled

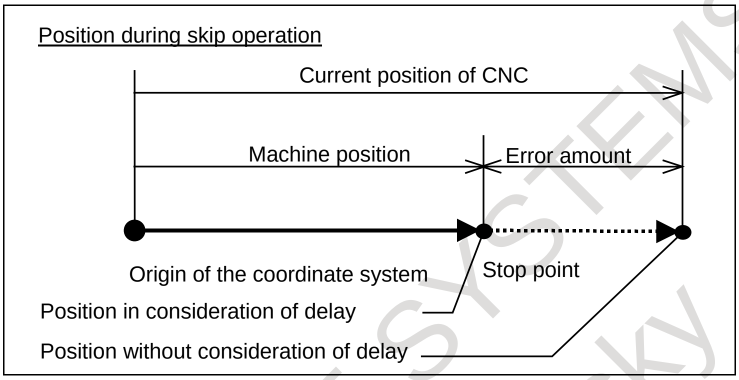
> ⚠ **CAUTION**
> 1 When bit 7 (SKF) of parameter No.6200 is set to 0, feedrate override, dry run, and automatic acceleration/deceleration are disabled in moving by the skip function regardless of the feedrate which is specified as a feed per minute or a feed per rotation.
> 2 Even if bit 7 (SKF) of parameter No.6200 is set to 1, dry run and automatic acceleration/deceleration are disabled when bit 1 (SFP) of parameter No.6207 is set to 1. When the multi-step skip is used and bit 2 (SFN) of parameter No.6207 is set to 1, dry run and automatic acceleration/deceleration are similarly disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
6201	**SKPXE**			**IGX**		**TSE**		

[Input type] Parameter input
[Data type] Bit path

#2 TSE When the torque limit skip function (G31 P99/98) is used, the skip position held in a system variable (#5061 to #5080 (#100151 to #100182 for a system having more than 20 axes)) is:

0: Position that is offset considering the delay (positional deviation) incurred by the servo system.
1: Position that does not reflect the delay incurred by the servo system.



#4 IGX When the high-speed skip function is used, SKIP, SKIPP, and SKIP2 to SKIP8 are:

0: Enabled as skip signals.
1: Disabled as skip signals.

#7 SKPXE For the skip function (G31), the skip signal SKIP is:

0: Enabled.
1: Disabled.

Whether the skip signals are enabled or disabled

Parameter	Bit 4 (IGX) of parameter No. 6201	Bit 0 (GSK) of parameter No. 6200	Bit 7 (SKPXE) of parameter No .6201	Skip signal SKIPP	Skip signal SKIP	Multistage skip signals SKIP2-SKIP8
Setting	0	0	0	Disabled	Enabled	Enabled
	0	1	0	Enabled	Enabled	Enabled
	0	0	1	Disabled	Disabled	Enabled
	0	1	1	Enabled	Disabled	Enabled
	1	0	0	Disabled	Disabled	Disabled
	1	1	0	Disabled	Disabled	Disabled
	1	0	1	Disabled	Disabled	Disabled
	1	1	1	Disabled	Disabled	Disabled

Bit 4 (IGX) of parameter No. 6201 is valid for the skip function using high-speed skip signals (when bit 4 (HSS) of parameter No. 6200 is set to 1) or for the multistage skip function using high-speed skip signals (when bit 5 (SLS) of parameter No. 6200 is set to 1).
To use multistage skip signals, the multistage skip function is required.

	#7	#6	#5	#4	#3	#2	#1	#0
6207						**SFN**	**SFP**	

[Input type] Parameter input
[Data type] Bit path

#1 SFP The feedrate used when the skip function (G31) is being executed is:
0: Feedrate of a programmed F code.
1: Feedrate set in parameter No. 6281.

> **NOTE**
> For the multi-stage skip function and high-speed skip, see the description of bit 2 (SFN) of parameter No. 6207.

#2 SFN The feedrate used when the skip function based on high-speed skip signals (with bit 4 (HSS) of parameter No. 6200 set to 1) or the multi-skip function is being executed is:
0: Feedrate of a programmed F code.
1: Feedrate set in a parameter from parameters Nos. 6282 to 6285.

> **NOTE**
> For not the multistage skip function, but the skip function using no high-speed skip signals (when bit 4 (HSS) of parameter No. 6200 is set to 0), see the description of bit 1 (SFP) of parameter No. 6207.

6281	**Feedrate for the skip function (G31)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets a feedrate for the skip function (G31). This parameter is valid when bit 1 (SFP) of parameter No. 6207 is set to 1.

> **NOTE**
> For the multi-stage skip function and high-speed skip, see the description of parameters Nos. 6282 to 6285.

6282	**Feedrate for the skip function (G31, G31 P1)**
6283	**Feedrate for the skip function (G31 P2)**
6284	**Feedrate for the skip function (G31 P3)**
6285	**Feedrate for the skip function (G31 P4)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Each of these parameters sets a feedrate for each skip function G code. These parameters are valid when bit 2 (SFN) of parameter No. 6207 is set to 1.

6221	Torque limit dead zone time for a torque limit skip command

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] 2msec
[Valid data range] 0 to 65535

The torque limit reached signal is ignored for a set period of time.
If G31P98 is specified, skip operation is not performed for a set period of time after the torque limit reached signal is set to “1”.
If G31P99 is specified, skip operation is not performed for a set period of time after the torque limit reached signal is set to “1”.
However, if a skip signal is input, skip operation is performed, regardless of the period of time set in this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
6286								TQO

[Input type] Parameter input
[Data type] Bit axis

#0 TQO The torque limit override function is:
0: Disabled. (Override of 100%)
1: Enabled.

> **NOTE**
> Before the torque limit skip function can be used, this parameter must be set to 1.

6287	Positional deviation limit in torque limit skip

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 327670

This parameter sets a positional deviation limit for each axis imposed when torque limit skip is specified. When the actual positional deviation exceeds the positional deviation limit, the alarm SV0004, “EXCESS ERROR (G31)” is issued and an immediate stop takes place.

Alarm and message

Number	Message	Description
PS0035	CAN NOT COMMANDED G31	- G31 cannot be specified. This alarm is generated when a G code (such as for cutter or tool-nose radius compensation) of group 07 is not canceled. - A torque limit skip was not specified in a torque limit skip command (G31P98 or P99). Specify the torque limit skip in the PMC window or the like. Or, specify the torque limit override by address Q.
PS0369	G31 FORMAT ERROR	- No axis is specified or two or more axes are specified in the torque limit switch instruction (G31P98/P99). - The specified torque Q value in the torque limit switch instruction is out of range. The torque Q range is 1 to 254.
SV0004	EXCESS ERROR (G31)	The amount of positional deviation during torque limit skip command operation exceeded the limit value of the parameter No. 6287.

Note

> **NOTE**
> This function is disabled for the synchronous slave axis and enabled for axes not related to composite control.

15.3.7 Skip Function of Measurement I/O Unit

Overview

Measurement I/O Unit is provided with an interface for connecting such measuring instruments as a touch probe for on-machine measurement system. It is possible to skip by the skip signals input from the Measurement I/O Unit. The skip signals are available for the G31 command and the measurement cycle. Up to two Measurement I/O Units can be connected and up to four skip signals can be input.

This function is effective to the following function:

- Skip function (G31)

Explanation

Skip signals of Measurement I/O Unit

When 1 Measurement I/O Unit is connected, 2 points SKIP_A and SKIP_B are available for the skip signals input.
When 2 Measurement I/O Units are connected, 4 points SKIP_A, SKIP_B, SKIP_C and SKIP_D are available for the skip signals input.

Correspondence between the number of connection and the skip signals is the following table.

Skip signal	Input of the unit	Unit connection order
SKIP_A	DI3A (pin11 of TM1)	1st unit
SKIP_B	DI3B (pin11 of TM2)	1st unit
SKIP_C	DI3A (pin11 of TM1)	2nd unit
SKIP_D	DI3B (pin11 of TM2)	2nd unit

The operation of all skip signals SKIP_A to SKIP_D are equivalent to that of the skip signal SKIP<X0004.7>.
The signals inputs of the skip signals SKIP_A to SKIP_D are always “1”.
The signals inputs are also “1” when bit 1 (SK0) of parameter No.6200 is set to 1.

To enable or disable skip signals

Bit 0 (MS1) to bit 3 (MS4) of parameter No.24714 enable to enable or disable each skip signals SKIP_A to SKIP_D.
Other parameters for selecting whether to enable or disable the skip signals are followings. However, the skip signals SKIP_A to SKIP_D are not affected by these parameters setting.

- bit 4 (IGX) of parameter No.6201
- bit 0 (GSK) of parameter No.6200
- bit 7 (SKPEXE) of parameter No.6201

Multi-step skip function

If the skip signals SKIP_A to SKIP_D are used for the multi-step skip or the dwell skip, please set bit 0 of parameters Nos. 6202 to 6206.

Delay time of skip signal

The parameters No.24715 to No.24718 enable to set the delay time of the skip signals SKIP_A to SKIP_D.
The coordinate values of the skip position are stored in the system variables for custom macros in consideration of the delay time.

Signal

Skip Signals SKIP_A to SKIP_D

[Classification] Input signal

[Function] This signal terminates skip cutting. That is, the position where a skip signal turns to 1 in a block containing G31 is stored in a custom macro variable, and the move command of the block is terminated at the same time.
Bit 0 (MS1) to bit 3 (MS4) of parameter No.24714 enable to enable or disable each skip signals SKIP_A to SKIP_D.

[Operation] When a skip signal turns to 1, the control unit works as described below.
When a block contains a skip cutting command G31, the control unit reads and stores the current position of the specified axis at that time. The control unit stops the axis, then cancels the remaining distance that the block was supposed to be moved.
There is no distinction between the skip signals SKIP_A to SKIP_D. The operation of all signals is equivalent to that of the skip signal SKIP<X0004.7>.

[Operation] When a skip signal turns to 1, the control unit works as described below.

- When a block contains a skip cutting command G31, the control unit reads and stores the current position of the specified axis at that time. The control unit stops the axis, then cancels the remaining distance that the block was supposed to be moved. There is no distinction between the skip signals SKIP_A to SKIP_D. The operation of all signals is equivalent to that of the skip signal SKIP<X0004.7>.

NOTE

1. The skip signal width requires at least 10 msec.
2. The delay or variation on the CNC side (excluding the PMC side) in detecting the skip signal is 0 to 2 msec. Therefore, the measurement error is the sum of 2 msec and the delay or variation (including the delay or variation of the receiver) of the skip signal on the PMC side, multiplied by the feedrate.
3. The delay or variation until feed stops after detection of the skip signal is 0 to 8 msec. To calculate the overshoot, further consider delay in acceleration/deceleration, delay in servo, delay on the PMC side.
4. The skip signals SKIP_A to SKIP_D do not need to be processed on the PMC side since the CNC directly reads the signals from the machine side.
5. When the skip signals SKIP_A to SKIP_D are monitored, the state rather than the rising edge is sampled. Therefore, when the level remains 1, the skip condition is assumed to be met immediately after skip cutting is specified next.
6. The skip signals SKIP_A to SKIP_D are effective to the EGB skip function and the Continuous high-speed skip function. When disabling them for each function, please set not to output the signals on the probe side.
7. In the case of the multi-path system, the skip signals SKIP_A to SKIP_D are common to all paths.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
24714					MS4	MS3	MS2	MS1

[Input type] Parameter input
[Data type] Bit path

MS1～MS4 This parameters enable to disable each skip signals SKIP_A to SKIP_D for Measurement I/O Unit. The table below indicates the correspondence between bits and skip signals. The setting of a bit has the following meaning:
0: Skip signal of Measurement I/O Unit corresponding to a bit is effective.
1: Skip signal of Measurement I/O Unit corresponding to a bit is not effective.

Parameter	Skip signal
MS1	SKIP_A
MS2	SKIP_B
MS3	SKIP_C
MS4	SKIP_D

24715	Delay time of skip signal 1 (Measurement I/O Unit)

[input type] Parameter input
[Data type] Word
[Unit of data] 0.1msec
[Valid data range] 0, 1 to 320
Delay time of skip signal SKIP_A for Measurement I/O Unit is set.
When setting is out of range, it is assumed that 0 is set.

NOTE
This parameter is effective to the following function.
- skip function (G31)

24716	Delay time of skip signal 2 (Measurement I/O Unit)

[input type] Parameter input
[Data type] Word
[Unit of data] 0.1msec
[Valid data range] 0, 1 to 320
Delay time of skip signal SKIP_B for Measurement I/O Unit is set.
When setting is out of range, it is assumed that 0 is set.

NOTE
This parameter is effective to the following function.
- skip function (G31)

24717	Delay time of skip signal 3 (Measurement I/O Unit)

[input type] Parameter input
[Data type] Word
[Unit of data] 0.1msec
[Valid data range] 0, 1 to 320
Delay time of skip signal SKIP_C for Measurement I/O Unit is set.
When setting is out of range, it is assumed that 0 is set.

> **NOTE**
> This parameter is effective to the following function.
> - skip function (G31)

24718	Delay time of skip signal 4 (Measurement I/O Unit)

[input type] Parameter input
[Data type] Word
[Unit of data] 0.1msec
[Valid data range] 0, 1 to 320

Delay time of skip signal SKIP_D for Measurement I/O Unit is set.
When setting is out of range, it is assumed that 0 is set.

> **NOTE**
> This parameter is effective to the following function.
> - skip function (G31)

15.4 COMPENSATION VALUE INPUT

T

15.4.1 Direct Input of Tool Offset Value Measured

Overview

This is a function of setting an offset value by key-inputting a workpiece dimension manually cut and measured from the MDI keyboard. First the workpiece is cut in the longitudinal or cross direction manually. When the position record signal is turned “1” (prepare a button on the machine operator’s panel) on completion of the cutting, the workpiece coordinate value of X axis(X axis of the basic three axes) and Z axis(Z axis of the basic three axes) at that time is recorded in the CNC. Then, withdraw the tool, stop the spindle, and measure the diameter or the radius if the cutting was on the longitudinal direction or measure the distance from the standard face if the cutting was on the facing. (The reference face is made as Z = 0.) When the measured value is entered on the offset value display screen, NC inputs the difference between the input measured value and the coordinate value recorded in NC, as the offset value of the offset number. If you release the tool without moving the tool in the axis along which an offset value is entered but moves the tool along the other axis, an offset value can be set without using the position record signal. The workpiece coordinate system can be shifted using the technique of directly inputting the measured value for offset. This technique is used when the coordinate system planned in the program does not match with the coordinate system actually set. The procedures are the same as those for direct input for offset, except a difference of using the standard tool on the work shift screen.
For details of the method of the operations, refer to the OPERATOR'S MANUAL (For Lathe System)(B-64694EN-1) of Setting and displaying data.

Explanation

- Relationship between the input of measurement value and the diameter/radius specification

Whether to set offset value by using a diameter value or radius value shift value depend on the setting of bit 3 (DIAx) of parameter No.1006 and bit 1 (ORC) of parameter No.5004.

Parameter DIAx(No.1006#3)	Parameter ORC(5004#1)	Offset value set
0	0	Radius
1	0	Diameter
0	1	Radius
1	1	Radius

Signal

Position record signal PRC <Gn040.6>

[Classification] Input signal

[Function] This signal is prepared for the function of direct input of tool offset value measured.
It is used to store in the control unit the data on the positions of the tool for tentative cutting. After measuring a dimension of the workpiece, input the measured value by the specified manual operation. The difference is then stored as the specified tool compensation value.

[Operation] The control unit stores the current position along X and Z axes when the signal turns to "1".

> ⚠ **CAUTION**
> To use this signal, set bit 2 (PRC) of parameter No. 5005 to 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn040		PRC						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5005						PRC		

[Input type] Parameter input
[Data type] Bit path

#2 PRC For direct input of a tool offset value or workpiece coordinate system shift amount:
0: The position record signal PRC <Gn040.6> is not used.
1: The position record signal PRC <Gn040.6> is used.

	#7	#6	#5	#4	#3	#2	#1	#0
11222			DRM					

[Input type] Parameter input
[Data type] Bit path

#5 DRM For direct input of tool offset value measured and direct input of coordinate system shift, whether to iput measurement value by using a diameter value or radius value depend on:
0: The setting of bit 3 (DIAx) of parameter No.1006.
1: The specification of dynamic switching of diameter/radius specification.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64694EN)	Setting and displaying data

15.4.2 Direct Input of Offset Value Measured B (for Lathe System)

T

Overview

When the touch sensor is provided, the tool offset value can be automatically settable in the tool offset memory, by moving the tool to make contact with the touch sensor during manual operation. The workpiece coordinate system shift amount can also be automatically set.

Explanation

- Touch sensor

Either of the following two cases may be selected depending on parameter setting.

1) If bit 3 (TS1) of parameter No. 5004 is 0
   The touch sensor has two direction–specific contact faces for each axis, thus outputting four signals (+MIT1 <X0004.2,X0013.2, X0011.2> ,+MIT2 <X0004.4,X0013.4,X0011.4> , -MIT1 <X0004.3,X0013.3,X0011.3> ,-MIT2 <X0004.5,X0013.5, X0011.5>) when contact is detected.
2) If bit 3 (TS1) of parameter No. 5004 is 1
   A touch sensor based on one contact input outputs one signal (+MIT1) when the one–contact input detects contact. So, which of directions of each axis is involved is automatically determined, and feeding in the corresponding axis direction is disabled.

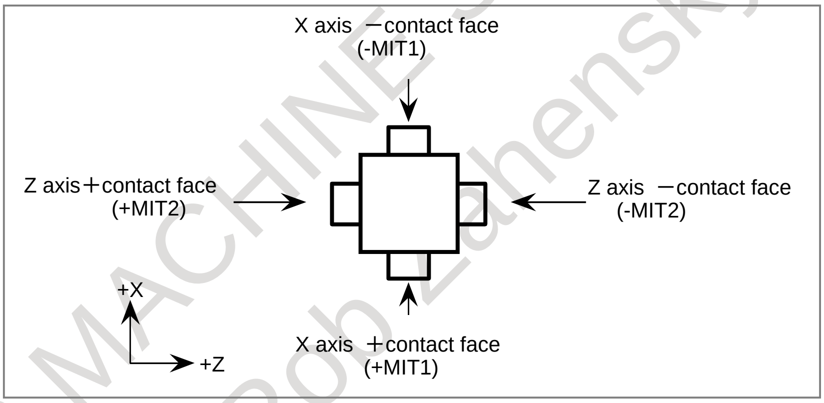


Fig. 15.4.2 (a) Touch Sensor

- Setting tool offset value

Determine a specific point on the machine tool as the measuring reference position. In advance, set the distance from this point to the measuring position (contact face of the touch sensor) as a reference value, using parameters Nos. 5015 to 5018. Select the tool whose offset value is to be measured, and bring it to touch the sensor, receiving a contact detection signal (tool compensation value write signal). The mechanical coordinate value is the distance from the tool nose position of the measuring tool at the mechanical reference (home) position to the measuring position; set the difference between this value and the reference value (parameter setting) into the tool offset value memory as the tool geometry offset value. The corresponding tool wear offset value becomes 0.

(Tool offset value to be set)

= (Mechanical coordinate value when tool compensation value write signal has become 1) – (Reference value (parameter value) corresponding to the tool compensation value write signal)

The tool geometry offset value to be set differs according to the method of determining the measuring reference position.

- If touch sensor contact detection is based on a one-contact input

If touch sensor contact detection is based on a one–contact input (bit 3 (TS1) of parameter No. 5004 is 1), when a contact detected signal (tool compensation amount write signal +MIT1) is received from the touch sensor, which of the two directions of each axis is involved is automatically determined according to several pulses stored as a result of the axis movement that was made before the signal reception. So the number of interpolation cycles related to the stored pulses must previously be set in parameter No. 5021. Once which of the two directions of each axis is involved is automatically determined, the corresponding axis direction is subjected to axis interlock to stop feeding, and the obtained tool geometry compensation amount is stored in tool compensation memory. If the directions of stored pulses are not unified, or no pulse has been stored, for example, because the servo power has been shut off (servo off) or no axis movement occurred, an alarm PS5195 is issued. An alarm PS5195is issued also if the tool moves along two axes (X–axis and Z–axis) simultaneously; move the tool along one axis at a time. If an alarm PS5195 is issued, no tool geometry compensation amount will be set up, resulting in the four directions (two axes) being subjected to interlock.

NOTE

1 Pulse storage for automatic decision is carried out in the manual mode while the GOSQM <Gn039.7> (tool compensation amount write mode select) signal is “1”. Stored pulses are lost if:
   a. The manual mode is exited,
   b. The GOSQM <Gn039.7> (tool compensation amount write mode select) signal becomes “0”,
   c. A contact detected signal is received from the touch sensor, resulting in a tool geometry compensation amount being set or an alarm PS5195 being issued,
   d. The servo power becomes off (resulting in the stored pulses for the related axis being lost), or
   e. Axis movement occurs (resulting in the stored pulses for the other axis being lost).

2 Axis interlock that has occurred for the axis direction identified by parameter–based automatic decision and two–axis, four–direction interlock that has occurred because of an alarm PS5195 being issued are canceled when the manual mode is exited or the GOSQM <Gn039.7> (tool compensation mount write mode select) signal becomes “0”.
   A reset does not cancel interlock.

- Determination of the axis movement direction in four-contact input

Set bit 3 (TS1) of parameter No. 5004 to 0 and bit 4 (TSD) of parameter No. 5009 to 1 to enable the input signals from the touch sensor and the movement direction of an axis to be monitored during measurement in the tool compensation write mode.
When the measurement is determined to be invalid, the moving axis is interlocked and an alarm PS5195 occurs.

- Conditions under which measurement is determined to be invalid
  The movement direction of an axis and the tool compensation write signals (+MIT1, +MIT2, -MIT1, -MIT2) are monitored in the tool compensation write mode (GOQSM<Gn039.7>=”1”). In the following case, the operation is determined to be invalid, the axis is interlocked, and an alarm PS5195 occurs.

  (1) A tool compensation write signal whose direction did not match the axis was input during measurement.
  (2) Any of the tool compensation write signals was input while the X- and Z-axes moved together.
  (3) Any of the tool compensation write signals was input without the axis being moved.

(4) The movement direction of the axis was not constant when the movement direction of the axis was determined (*1).

(*1) When the axis moves in one direction of one axis by the compensation cycle count set in parameter No. 5021, the axis is assumed to move in the direction.

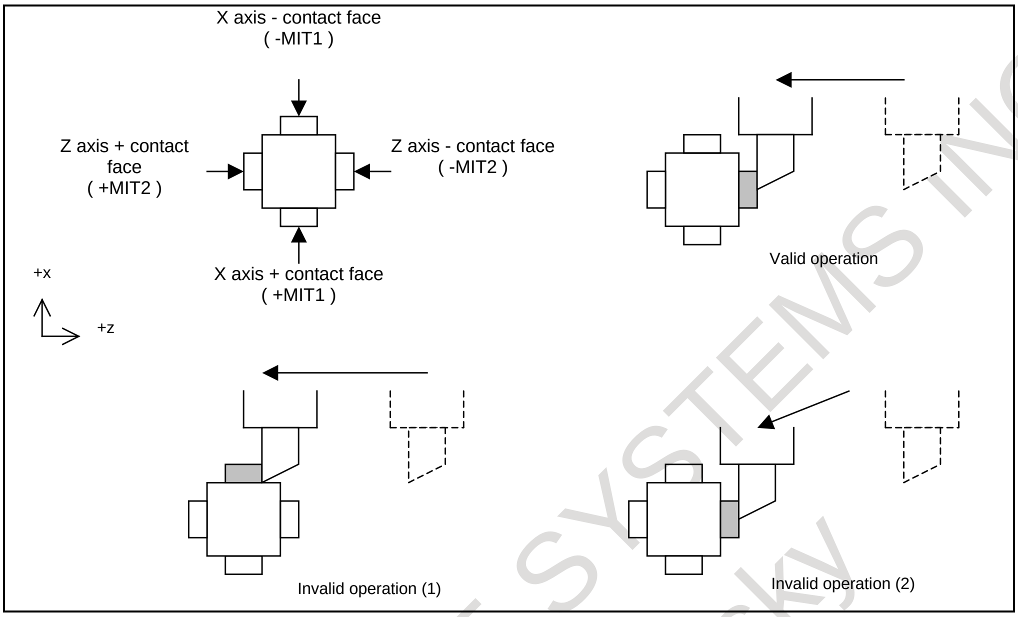


Fig. 15.4.2 (b) Examples of operations determined to be invalid

- Interlock applied when measurement is invalid
  When measurement is determined to be invalid, movement only in the direction opposite to the previous movement direction is allowed. That is, the directions other than the direction opposite to the previous movement direction are interlocked.

  Examples of interlock
  (1) When a tool compensation write signal other than Z-axis/contact surface (-MIT2) is input during movement in the minus direction of the Z-axis, the minus direction of the Z-axis and the plus and minus directions of the X-axis are interlocked.
  (2) When any of tool compensation write signals is input during movement in the plus direction of the X-axis and the minus direction of the Z-axis, the plus directions of the X-axis and the minus direction of the Z-axis are interlocked.
  (3) When any of tool compensation write signals is input without axis movement, the four directions of the two axes are interlocked.

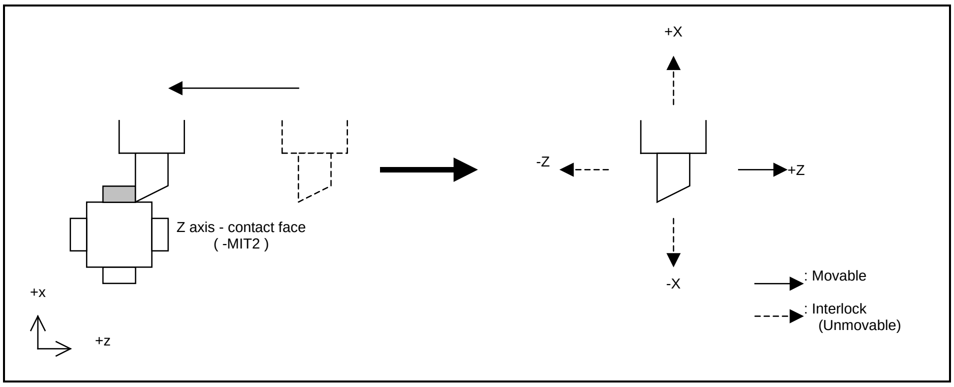


Fig. 15.4.2 (c) Interlock direction in example (1)

The release interlock applied because measurement is determined to be invalid, keep the tool away from the touch sensor (+MIT1, +MIT2, -MIT1, -MIT2 = “0”) and reset the CNC. If the tool is not kept away from the touch sensor, resetting the CNC does not release interlock.
In conditions (3) and (4) in which measurement is determined to be invalid, no axis can be moved because the four directions of the two axes are interlocked. In such a case, cancel the tool compensation write mode and reset the CNC to release the interlock. To cancel the tool compensation write mode, set the tool compensation write mode signal GOQSM <Gn039.7> is to “0” or select the MDI mode, MEM mode, or EDIT mode.
A reset releases interlock and an alarm. Even if another tool compensation write signal (+MIT1, +MIT2, -MIT1, or -MIT2) is input before the interlock or alarm is released, the signal is ignored.

- For direct input of offset value measured B for two-spindle lathe

When the tool compensation number specification signal ONSC <Gn547.6> is to “0”, parameter No. 5053 is used to specify for which spindle (1 or 2) tool compensation numbers (tool compensation amount memory) are to be used.

(Example)
When there are 32 tool compensation pairs

Tool compensation number	Parameter No. 5053	
	8	10
Spindle 1	1 to 8	1 to 10
Spindle 2	9 to 32	11 to 32

NOTE
If parameter No. 5053 for grouping tool compensation numbers is 0 or greater than the maximum allowable number of tool compensation sets, grouping is carried out as listed below.

Number of tool compensation pairs	32 pairs	64 pairs	99 pairs	200 pairs
Spindle 1	1 to 16	1 to 32	1 to 49	1 to 100
Spindle 2	17 to 32	33 to 64	50 to 98	101 to 200

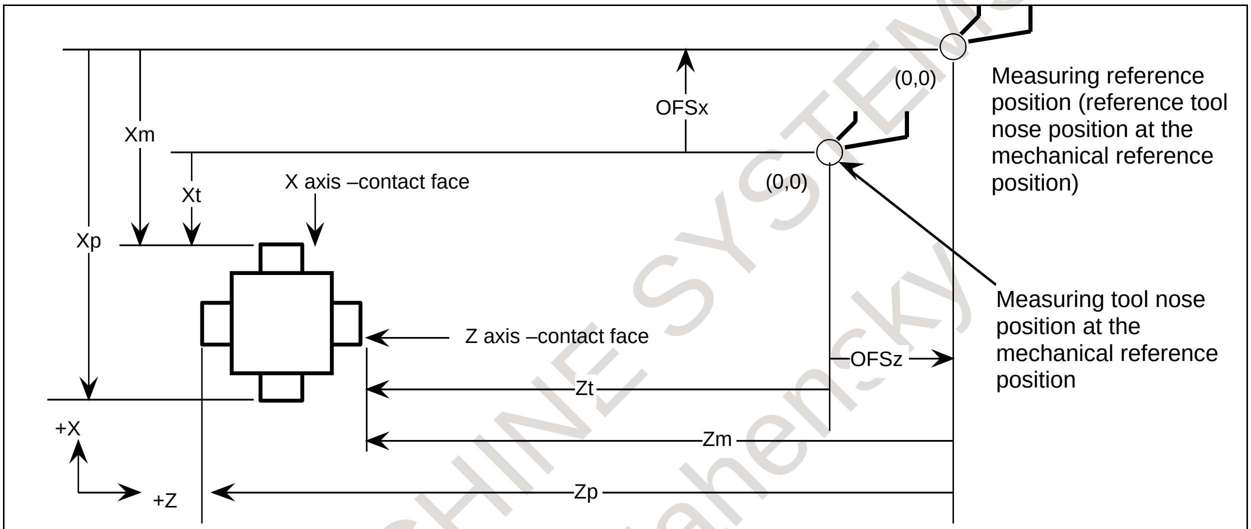
Number of tool compensation pairs	400 pairs	499 pairs	999 pairs
Spindle 1	1 to 200	1 to 249	1 to 499
Spindle 2	201 to 400	250 to 498	500 to 998

When tool compensation number specification signal ONSC is “1”, the tool compensation amount memory is shared between spindle 1 and spindle 2 regardless of the setting of parameter No. 5053. Spindle measurement selection signal S2TLS <Gn040.5> is used only to select which spindle is measured; any tool compensation number within the number of tool compensation pairs can be selected for each spindle.
(Example)
When the following settings are made, the tool geometry compensation number to be selected is shown in the table below.
Number of tool compensation pairs is 32.
Bit 5 (QNI) of parameter No. 5005 = 1:
The tool compensation number is set by signal input from the PMC.
Parameter No. 5053 = 10:
Tool compensation number shift amount
Tool offset number selection signals OFN0 to OFN9=2:
A tool compensation number of 3 is specified.

Tool compensation number specification signal ONSC	Spindle measurement select signal S2TLS	Tool compensation number to be selected
0	0	3
0	1	13
1	0	3
1	1	3

Example 1

The difference between the reference tool nose tip position and the measuring tool nose tip position can be set as the tool offset value. Define the reference tool nose tip position at the mechanical reference position (machine zero position) as the measuring reference position, then set the distances Xp, Zp, Xm, Zm, from the measuring reference position to the contact faces of the sensor in parameters.



Xp: Distance from the measuring reference position to X-axis (+) contact face (parameter No.5015)
Xm: Distance from the measuring reference position to X-axis (–) contact face (parameter No.5016)
Zp: Distance from the measuring reference position to Z-axis (+) contact face (parameter No.5017)
Zm: Distance from the measuring reference position to Z-axis (–) contact face (parameter No.5018)
Xt: X-axis direction moving distance of the measuring tool up to the contact face of sensor (X-axis machine coordinate value)
Zt: Z-axis direction moving distance of the measuring tool up to the contact face of sensor (Z-axis machine coordinate value)
(when Xt and Zt touch the X-axis (–) contact face and Z-axis (–) contact face in the above figure)
OFSx: Tool offset value to be set (X-axis): OFSx = Xt - Xm
OFSz: Tool offset value to be set (Z-axis): OFSz = Zt - Zm

Fig. 15.4.2 (d) When the reference tool nose tip position is set as the measuring reference position

When two touch sensors are used (Parameter 2NR (No.5051#0)=1), each parameter is set as follows.

Content	For touch sensor 1	For touch sensor 2
Distance to X-axis + contact surface of touch sensor	5015	5056
Distance to X-axis - contact surface of touch sensor	5016	5057
Distance to Z-axis + contact surface of touch sensor	5017	5058
Distance to Z-axis - contact surface of touch sensor	5018	5059

Example 2

The measuring reference point may be an imaginary point (imaginary zero point), as shown in the Fig. 15.4.2 (e). The difference between the imaginary zero point and the measuring tool nose tip position at the mechanical reference point can be set as the tool geometry offset value of the measuring tool, by setting the distances from the imaginary zero point to the respective contact faces in parameters.

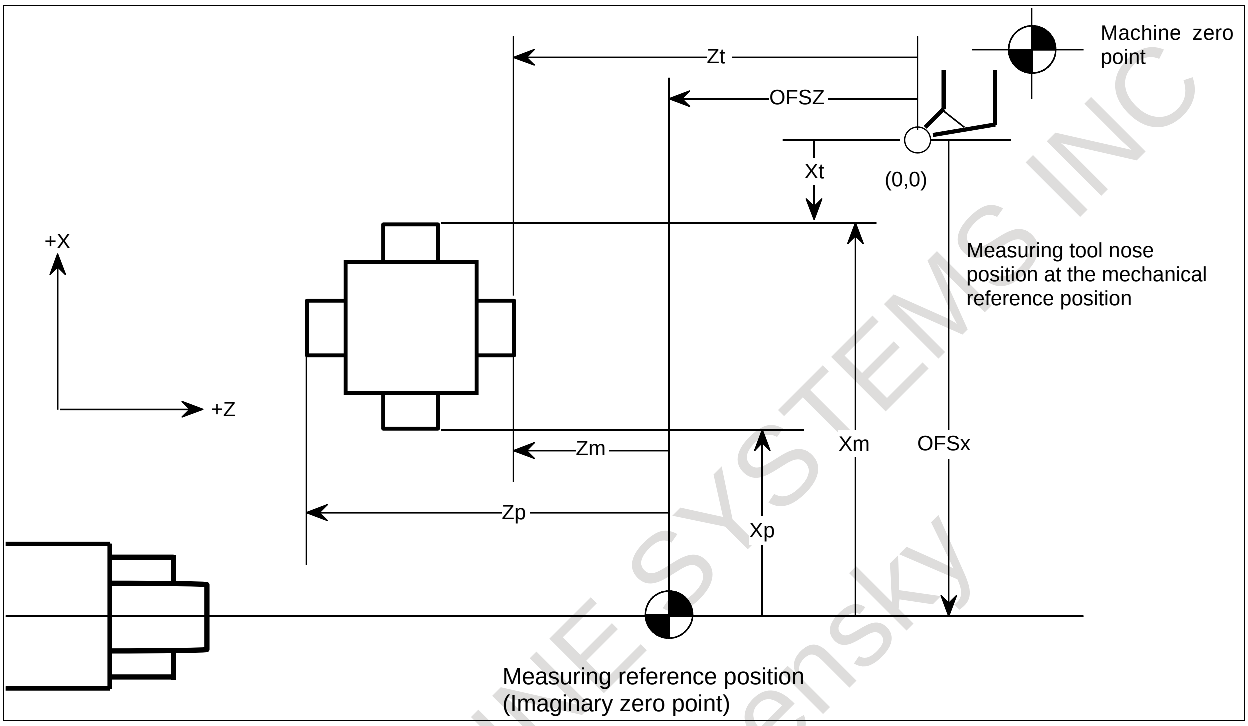


Fig. 15.4.2 (e) When the imaginary zero position is set as the measuring reference position

- Setting the workpiece coordinate system shift amount

The workpiece coordinate system shift amount for the Z-axis can be set as follows: Bring the tool to touch the workpiece end face. Subtract the tool geometry offset value of the tool (the value shifted in the coordinate system by the tool geometry offset) from the machine coordinate value (the distance from the measuring tool nose tip position at the mechanical reference position (machine zero point) to the workpiece end face).The result is set as the workpiece coordinate system shift value.

[Z axis workpiece coordinate system shift amount to be set (EXOFSz)]

= [Z axis tool geometry offset value of the corresponding tool (OFSz)] – [Z axis machine coordinate value(Zt)]

Using the above methods, the workpiece coordinate system is set with the workpiece end face (the contact point of the sensor) specified as the programmed zero point of the workpiece coordinate system of the Z-axis.

- If the direct input of offset value measured B for two-spindle lathe is used

  For the direct input of offset value measured B for two-spindle lathe, the workpiece coordinate shift amount for the Z–axis is automatically set in any of workpiece coordinate systems G54 to G59 for workpiece coordinate system memory.

  [Z–axis workpiece coordinate system shift amount to be set up]

  = –([corresponding Z–axis tool geometry compensation amount] – [Z–axis machine coordinate value])

  Select the desired workpiece coordinate systems from G54 to G59 manually or automatically, using the 2AT parameter (bit 1 of No. 5051).

  - 2AT = 0 (manual selection)

    Set the cursor to the desired workpiece coordinate system G54 to G59
    by operating the MDI (such as page and cursor keys).

> **NOTE**
> It is inhibited to select an external workpiece coordinate system offset (EXT). If the cursor is set to it, the warning message ”Write inhibited” is displayed when the workpiece coordinate system shift write amount signal becomes “1”.

- 2AT = 1 (automatic selection)
  Previously set any of workpiece coordinate systems G54 to G59 in parameter No. 5054 for spindle 1 and parameter No. 5055 for spindle 2. When the workpiece coordinate system shift amount write mode is entered, the cursor is set to any of workpiece coordinate systems G54 to G59.

> **NOTE**
> If 0 or a value out of the valid data range is set in parameter No. 5053 or 5054, workpiece coordinate systems G54 and G57 are set up, respectively, for spindles 1 and 2.

Also select a tool compensation number to be used in measurement manually or automatically, using the bit5 (QNI) of parameter No.5005.

- QNI = 0 (manual selection)
  Specify the desired tool compensation number in parameter No. 5020.
- QNI = 1 (automatic selection)
  The cursor is set to the desired tool compensation number, using the tool compensation number input signal.
  For spindle 1, the cursor is set to a tool compensation number selected by the input signal.
  For spindle 2, the cursor is set to the sum of the value specified in parameter No. 5053 and the tool compensation number selected by the input signal is used.

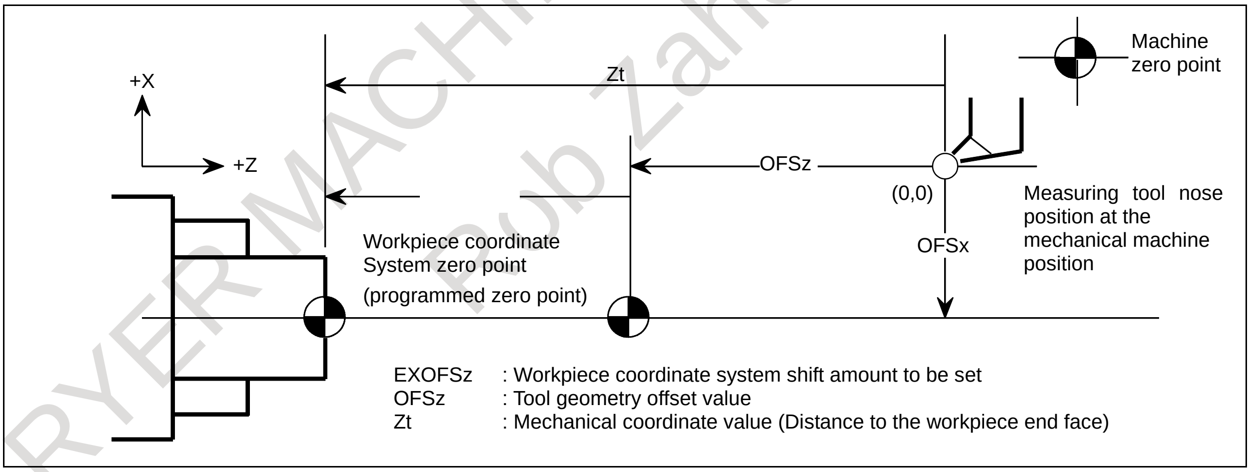
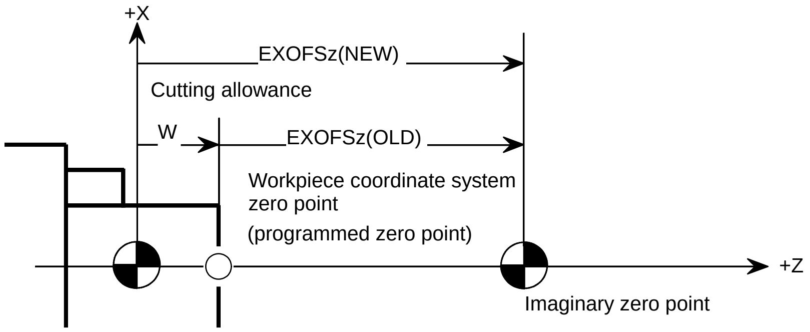


Fig. 15.4.2 (f) Setting of workpiece coordinate system shift amount

To deviate the programmed zero point of the workpiece coordinate system from the workpiece end face, such as by adding a cutting allowance, use the incremental input of the workpiece coordinate system shift amount in MDI operation. By setting the distance from the programmed zero point to the workpiece end face with a sign, the numeric value input is added to the preset amount.



EXOFSz(OLD) : Workpiece coordinate system shift amount being set by the function of input of tool offset value measured B
W : Cutting allowance (incremental input value)
EXOFSz(NEW) : Workpiece coordinate system shift amount after adding the cutting allowance

Fig. 15.4.2 (g) Setting of cutting allowance

- Basic procedure to set tool offset value

To use the direct input of offset value measured B for two-spindle lathe, first specify the spindle to be measured, using the S2TLS (spindle measurement select) signal.

(1) Execute manual reference position return.
By executing manual reference position return, a machine coordinate system is established.
The tool offset value is computed on the machine coordinate system.

(2) Select manual handle mode or manual continuous feed mode and set the tool compensation value write mode select signal GOQSM to “1”. The display is automatically changed to the tool offset screen (geometry), and the “OFST” indicator starts blinking in the status indication area at the bottom of the screen, which indicates that the tool compensation value writing mode is ready. When the direct input of offset value measured B for two-spindle lathe is in use, the S1MES <Fn062.3> or S2MES <Fn062.4> (spindle under measurement) signal, whichever is applicable, becomes “1”.

NOTE
After this, do not change the S2TLS (spindle measurement selection) signal until the GOQSM (offset write mode) signal becomes 0.

(3) Select a tool to be measured.

(4) When the cursor does not coincide with the tool offset number desired to be set, move the cursor to the desired tool offset number using the page key and cursor key.
The cursor can also be coincided with the tool offset number desired to be set automatically by the tool offset number input signals (when bit 5 (QNI) of parameter No. 5005=1).
In this case, the position of the cursor cannot be changed on the tool compensation screen using page keys or cursor keys.

(5) Near the tool to the sensor by manual operation.

(6) Place the tool edge to a contacting surface of the sensor by manual handle feed.
Bring the tool edge in contact with the sensor. This causes the tool compensation value writing signals to input to be CNC.
The following tool compensation amount write signals are set up according to the setting of the TS1 parameter (bit 3 of No. 5004).
When the parameter is 0: +MIT1, –MIT1, +MIT2, –MIT2
When the parameter is 1: +MIT1 only
The tool compensation value writing signal is set to “1”, and then :
i) The axis is interlocked in this direction and its feed is stopped.
ii) The tool offset value extracted by the tool offset memory (tool geometry offset value) which corresponds to the tool offset number shown by the cursor is set up.

(7) For both X-axis and Z-axis, their offset values are set by operations (5) and (6).

(8) Repeat operations (3) to (7) for all necessary tools.

(9) Set the tool compensation value writing mode signal GOQSM to “0”.
The writing mode is canceled and the blinking “OFST” indicator light goes off.

When the direct input of offset value measured B for two-spindle lathe is in use, the S1MES or S2MES (spindle under measurement) signal for the spindle being measured becomes 0.

- Basic procedure to set workpiece coordinate shift value

To use the direct input of offset value measured B for two-spindle lathe, first specify the spindle to be measured, using the S2TLS (spindle measurement select) signal.

(1) Set the tool geometry offset values of each tool in advance.

(2) Execute manual reference position return.
By executing manual reference position return, the machine coordinate system is established.
The workpiece coordinate system shift amount is computed based on the machine coordinate system of the tool.

(3) Set the workpiece coordinate system shift amount writing mode select signal WOQSM <Gn039.6> to “1”.
The display automatically switches to the workpiece shifting screen, the “WFST” indicator starts blinking at the status indicator area in the bottom of the screen, which indicates that the workpiece coordinate system shift amount writing mode is ready.
When the direct input of offset value measured B for two-spindle lathe is in use, the workpiece coordinate system screen is selected, and the S1MES or S2MES (spindle under measurement) signal, whichever is applicable, becomes “1”.

> **NOTE**
> After this, it is impossible to switch the S2TLS (spindle measurement selection) signal until the WOQSM (offset write mode) signal becomes “0”.

(4) Select a tool to be measured.

(5) Check tool offset numbers.
The tool offset number corresponding to the tool required for measurement, shall be set in the parameter No. 5020 in advance.
The tool offset number can also be set automatically by setting the tool offset number input signal (with bit 5 (QNI) of parameter No. 5005=1).

(6) Manually approach the tool to an end face of the workpiece.

(7) Place the tool edge to the end face (sensor) of the workpiece using manual handle feed.
When the tool edge contacts the end face of the workpiece, input the workpiece coordinate system shift amount signal WOSET <Gn040.7>.
The workpiece coordinate system shift amount on the Z–axis is automatically set.

(8) Release the tool.

(9) Set the workpiece coordinate system shift amount write mode select signal WOQSM to “0”.
The writing mode is canceled and the blinking “WSFT” indicator light goes off.
When the direct input of offset value measured B for two-spindle lathe is in use, the S1MES or S2MES (spindle under measurement) signal, whichever is applicable, becomes “0”.

- Switching from a customized user screen to the measurement screen

When the MANUAL GUIDE i screen or a user screen customized by C Language Executor or macro executor is displayed, if tool compensation amount write mode select signal GOQSM or workpiece coordinate system shift amount write mode signal WOQSM is set to 1, a shift to the measurement screen is made automatically.

When a user screen customized by C Language Executor is assigned as the tool geometry compensation screen or workpiece shift screen, turning on the measurement mode switches to the user screen, not to the CNC screen.

Limitation

This function cannot be used with the following functions.

- Composite control
- Mirror image
- Programmable mirror image
- Mirror image for double turret

The direct input of offset value measured B for two-spindle lathe function is also the same.

Signal

Tool offset write mode select signal GOQSM <Gn039.7>

[Classification] Input signal

[Function] Select the mode for writing tool compensation

[Operation] When this signal is turned “1” in a manual operation mode, the mode for writing tool compensation is selected. The control unit then automatically switches the screen on the display to the tool geometry compensation screen and blinks the “OFST” status display at the bottom of the screen to notify that the mode has been changed to the mode for writing tool compensation.

Tool offset write signals
(Path 1) +MIT1#1, +MIT2#1<X0004.2, X0004.4> -MIT1#1, -MIT2#1<X0004.3, X0004.5>
(Path 2) +MIT1#2, +MIT2#2<X0013.2, X0013.4>-MIT1#2, -MIT2#2<X0013.3, X0013.5>
(Path 3) +MIT1#3, +MIT2#3<X0011.2, X0011.4>-MIT1#3, -MIT2#3<X0011.3, X0011.5>

[Classification] Input signal

[Function] Each of these signals inhibits the tool from being fed along the corresponding axis during manual operation. When signal GOQSM for selecting the mode for writing tool compensation is set to “1”, the manual feed is inhibited and also the tool geometry compensation along the axis is automatically calculated and the result is set in tool compensation memory.

[Operation] When these signals are turned “1” during tool offset write mode, the control unit operates as follows:

- Inhibits tools from being fed along the corresponding axis during manual operation.
  If bit 3 (TS1) of parameter No. 5004 is set to 0
  +MIT1 : Inhibits the tool from being manually fed in the positive direction along the X-axis.
  –MIT1 : Inhibits the tool from being manually fed in the negative direction along the X-axis.
  +MIT2 : Inhibits the tool from being manually fed in the positive direction along the Z-axis.
  –MIT2 : Inhibits the tool from being manually fed in the negative direction along the Z-axis.
  If bit 3 (TS1) of parameter No. 5004 is set to 1
  +MIT1 : Automatic decision causes the related two directions of each axis to be subjected to interlock.
  –MIT1 : Not used
  +MIT2 : Not used
  –MIT2 : Not used
- When signal GOQSM for selecting the mode for writing tool compensation is turned “1”, the interlock signal also automatically calculates the tool geometry compensation for the tool compensation number pointed to by the cursor and sets the result in tool compensation memory.

> **NOTE**
> 1 This signal is used as the interlock signal in each axis direction.
> 2 This signal is valid only when the bit 0 (GSC) of parameter No. 5009 is 0.
> 3 In case of bit 2 (XSG) of parameter No. 3008=1, the address assigned by parameter No. 3019 is used

---

Tool offset write signals
+MIT1to +MIT2 <Gn132.0 to Gn132.1>, -MIT1 to-MIT2 <Gn134.0 to Gn134.1>

[Classification] Input signal

[Function] Each of these signals inhibits the tool from being fed along the corresponding axis during manual operation. When signal GOQSM for selecting the mode for writing tool compensation is set to “1”, the manual feed is inhibited and also the tool geometry compensation along the axis is automatically calculated and the result is set in tool compensation memory.

[Operation] When these signals are turned “1”, the control unit operates as follows:

- Inhibits tools from being fed along the corresponding axis during manual operation.
  When bit 3 (TS1) of parameter No. 5004 is 0 and the X-axis is assumed to be the first axis and the Z-axis to be the second axis
  - +MIT1 : Inhibits the tool from being manually fed in the positive direction along the X-axis.
  - –MIT1 : Inhibits the tool from being manually fed in the negative direction along the X-axis.
  - +MIT2 : Inhibits the tool from being manually fed in the positive direction along the Z-axis.
  - –MIT2 : Inhibits the tool from being manually fed in the negative direction along the Z-axis.

  If bit 3 (TS1) of parameter No. 5004 is set to 1
  - +MIT1 : Automatic decision causes the related two directions of each axis to be subjected to interlock.
  - –MIT1 : Not used
  - +MIT2 : Not used
  - –MIT2 : Not used
- When signal GOQSM for selecting the mode for writing tool compensation is turned “1”, the interlock signal also automatically calculates the tool geometry compensation for the tool compensation number pointed to by the cursor and sets the result in tool compensation memory.

> **NOTE**
> This signal is valid only when the bit 0 (GSC) of parameter No. 5009 is 1.

Tool offset number selection signals
OFN0 to OFN5, OFN6 to OFN9 <Gn039.0 to Gn039.5, Gn040.0 to Gn040.3>

[Classification] Input signal

[Function] Select the tool offset number.

[Operation] When the mode for writing tool compensation is selected, the cursor is automatically positioned on the tool geometry compensation number selected by these signals.
A tool offset number is specified by a 10–bit binary number. Numbers 0 to 998 correspond to the compensation number 1 to 999.

NOTE
This signal is available only when bit 5 (QNI) of parameter No. 5005 =1.

Workpiece coordinate system shift value write mode select signal WOQSM <Gn039.6>

[Classification] Input signal

[Function] Select the mode for writing the shift amount for the workpiece coordinate system.

[Operation] When this signal is turned to “1” in a manual operation mode, the mode for writing the shift amount for the workpiece coordinate system is selected. The control unit then automatically switches the screen displayed to the WORK SHIFT screen and blinks the “OFST” status display at the bottom of the screen to notify that the mode has been changed to the mode for writing the shift amount for the workpiece coordinate system. However, this is not performed when the mode for writing tool compensation values is selected.

Spindle measurement select signal S2TLS <Gn040.5>

[Classification] Input signal

[Function] For the direct input of offset value measured B for two-spindle lathe, a spindle is selected for measurement. If the signal is “0“, spindle 1 is selected. If it is “1“, spindle 2 is selected.
If spindle 2 is to be selected for measurement, the tool offset write mode select signal GOQSM and the workpiece coordinate system shift value write mode select signal WOQSM as well as the spindle measurement select signal must be set to 1.

[Operation] If the signal is “0“, spindle 1 is selected. If it is “1“, spindle 2 is selected.

Workpiece coordinate system shift value write signal WOSET <Gn040.7>

[Classification] Input signal

[Function] Automatically calculates and sets the shift amount for the workpiece coordinate system.

[Operation] When this signal turns to “1“ in the mode for writing the shift amount for the workpiece coordinate system, it triggers the automatic calculation and setting of the shift amount for the workpiece coordinate system.

Tool compensation number specification signal ONSC <Gn547.6>

[Classification] Input signal

[Function] When bit 5 (QNI) of parameter No. 5005 is 1 in direct input of offset value measured B for two-spindle lathe, the tool compensation number specified by tool offset number selection signals OFN0 to OFN9 is selected as is, regardless of spindle measurement select signal S2TLS and the setting of parameter No. 5053.

[Operation] When this signal is set to "1", the tool compensation number specified by tool offset number selection signals OFN0 to OFN9 is selected as is, regardless of spindle measurement select signal S2TLS and the setting of parameter No. 5053.

Spindle 1 under measurement signal S1MES <Fn062.3>
Spindle 2 under measurement signal S2MES <Fn062.4>

[Classification] Output signal

[Function] For the direct input of offset value measured B for two-spindle lathe, it is indicated which spindle, 1 or 2, is under measurement.

[Operation] These signals check which head is being measured in the tool offset write mode or workpiece coordinate system shift amount write mode according to the S2TLS (spindle measurement select) signal.
If spindle 1 is selected, S1MES = “1“ and S2MES = “0“. If spindle 2 is selected, S1MES = “0“ and S2MES = “1“.

Signal address

(Path 1)

	#7	#6	#5	#4	#3	#2	#1	#0
X0004			-MIT2	+MIT2	-MIT1	+MIT1		
			SKIP5	SKIP4	SKIP3	SKIP2		

(Path 2)

	#7	#6	#5	#4	#3	#2	#1	#0
X0013			-MIT2[#2]	+MIT2[#2]	-MIT1[#2]	+MIT1[#2]		
			SKIP5[#2]	SKIP4[#2]	SKIP3[#2]	SKIP2[#2]		

(Path 3)

	#7	#6	#5	#4	#3	#2	#1	#0
X0011			-MIT2[#3]	+MIT2[#3]	-MIT1[#3]	+MIT1[#3]		
			SKIP5[#3]	SKIP4[#3]	SKIP3[#3]	SKIP2[#3]		

> ⚠ **CAUTION**
> Since the same addresses are used for both +MIT1, MIT1,+MIT2, –MIT2 and skip signals SKIP2 to SKIP5 (multi–step skip), be careful when using these two signal types.

	#7	#6	#5	#4	#3	#2	#1	#0
Gn039	GOQSM	WOQSM	OFN5	OFN4	OFN3	OFN2	OFN1	OFN0

	#7	#6	#5	#4	#3	#2	#1	#0
Gn040	WOSET		S2TLS		OFN9	OFN8	OFN7	OFN6

	#7	#6	#5	#4	#3	#2	#1	#0
Gn132							+MIT2	+MIT1

	#7	#6	#5	#4	#3	#2	#1	#0
Gn134							-MIT2	-MIT1

	#7	#6	#5	#4	#3	#2	#1	#0
Gn547		ONSC						

	#7	#6	#5	#4	#3	#2	#1	#0
Fn062				S2MES	S1MES			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3003					DIT			

[Input type] Parameter input
[Data type] Byte path

#3 DIT The interlock signal for each axis direction is:
0: Valid.
1: Invalid.

	#7	#6	#5	#4	#3	#2	#1	#0
3008						XSG		

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#2 XSG A signal assigned to an X address is:
0: Fixed at the address.
1: Able to be reassigned to an arbitrary X address.

NOTE
When this parameter is set to 1, set parameters Nos. 3013, 3014, 3012, and 3019. If parameters Nos. 3013 and 3014 are not set, the deceleration signal for reference position return is assigned to bit 0 of <X0000>. If parameters Nos. 3012 and 3019 are not set, the skip signal, the PMC axis control skip signal, the measurement position arrival signal, the interlock signal for each axis direction, and the tool compensation value write signal are assigned to <X0000>.

⚠ WARNING
Set an appropriate value to parameters (Nos. 3013, 3014, 3012, 3019) beforehand when you set 1 to this parameter.
If an input signal of the set address in parameters (Nos. 3013, 3014, 3012, 3019) is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used X address.

3019	Address to which the PMC axis control skip signal, measurement position arrival signal, and tool offset write signals are assigned

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 727

Set an X address to which the PMC axis control skip signal ESKIP, measurement position arrival signals (XAE, YAE, and ZAE (M series) or XAE and ZAE (T series)), and tool offset write signals (±MIT1 and ±MIT2 (T series)) are to be assigned.

Example 1. When parameter No. 3012 is set to 5 and parameter No. 3019 is set to 6

When bit 2 (XSG) of parameter No. 3008 is 1, the PMC axis control skip signal, and measurement position arrival signal are allocated to X0006 and the skip signal is allocated to X0005.

	#7	#6	#5	#4	#3	#2	#1	#0	
X0005	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	(T series)
	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	(M series)

	#7	#6	#5	#4	#3	#2	#1	#0	
X0006		ESKIP	-MIT2	+MIT2	-MIT1	+MIT1	ZAE	XAE	(T series)
		ESKIP				ZAE	YAE	XAE	(M series)

Example 2. When parameter No. 3012 is set to 5 and parameter No. 3019 is set to 5

When bit 2 (XSG) of parameter No. 3008 is 1, the PMC axis control skip signal, measurement position arrival signal, and skip signal are allocated to X0005.

X0005	#7	#6	#5	#4	#3	#2	#1	#0	
	SKIP	ESKIP	-MIT2	+MIT2	-MIT1	+MIT1	ZAE	XAE	(T series)
		SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	

	#7	#6	#5	#4	#3	#2	#1	#0	
	SKIP	ESKIP	SKIP5	SKIP4	SKIP3	ZAE	YAE	XAE	(M series)
		SKIP6				SKIP2	SKIP8	SKIP7	

NOTE

1 This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
Depending on the configuration of the I/O Link, the actually usable X addresses are:
<X0000 to X0127>, <X0200 to X0327>, <X0400 to X0527>, <X0600 to X0727>

2 When this parameter is 0, an input signal from address X0 is used.

⚠ **WARNING**

If an input signal of the set address in this parameter is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used X address.

	#7	#6	#5	#4	#3	#2	#1	#0
5004					TS1			

[Input type] Parameter input
[Data type] Bit path

#3 TS1 For touch sensor contact detection with the function for direct input of offset value measured B:
0: Four-contact input is used.
1: One-contact input is used.

NOTE

For the machining center system, set TS1 to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
5005		TLE	QNI					

[Input type] Parameter input
[Data type] Bit path

#5 QNI With the tool length/workpiece zero point measurement function or the function for direct input of offset value measured B, a tool compensation number is selected by:
0: Operation through the MDI panel by the operator (selection based on cursor operation).
1: Signal input from the PMC.

#6 TLE The "direct input of offset value measured B" function updates the offset value in offset write mode:
0: Constantly.
1: During axis movement.
Axis movement assumes a positional deviation other than 0.

	#7	#6	#5	#4	#3	#2	#1	#0
5009				TSD				GSC

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

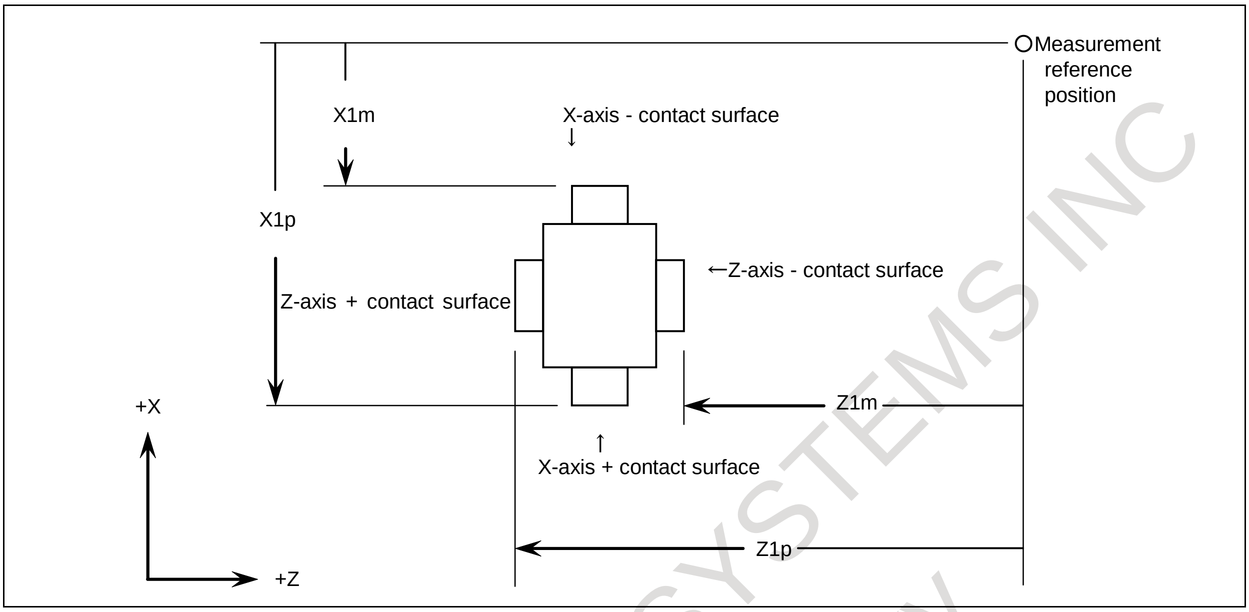
#0 GSC When the function for direct input of offset value measured B is used, an offset write input signal is input from:
0: Machine side
1: PMC side
When the interlock function for each axis direction is enabled (when bit 3 (DIT) of parameter No. 3003 is set to 0), switching can also be made between input from the machine side and input from PMC side for the interlock function for each axis direction.

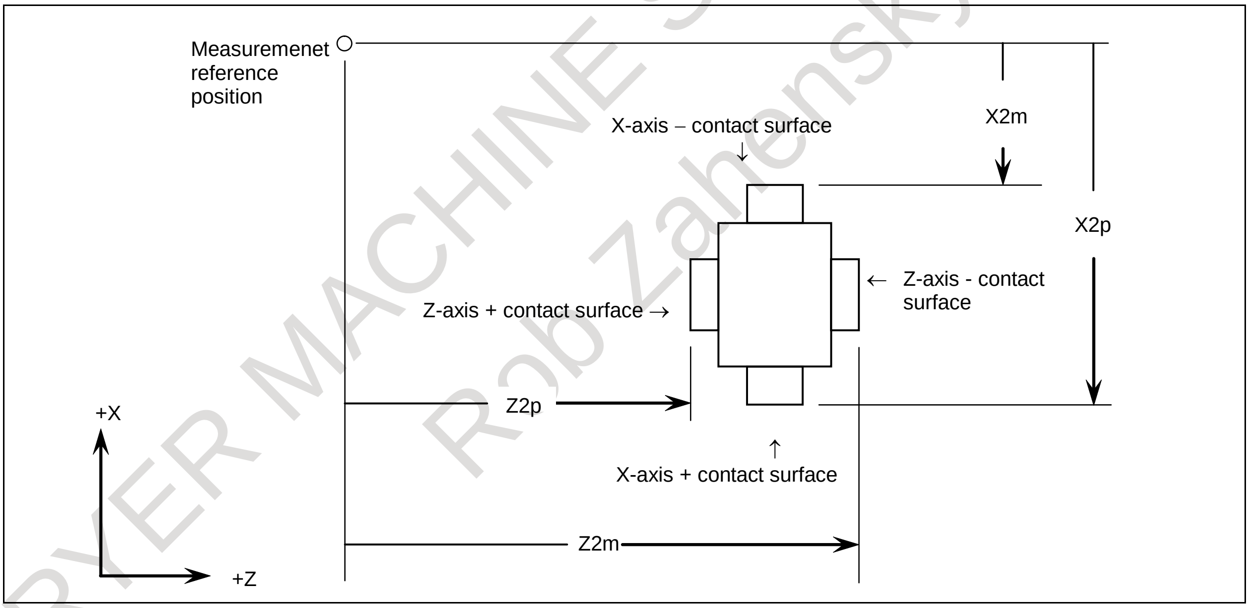
#4 TSD In the function for direct input of offset value measured B, the movement direction determination specifications:
0: Do not apply.
1: Apply.
This parameter is valid when four-contact input is used (bit 3 (TS1) of parameter No. 5004 is set to 0).

5015	Distance to X-axis + contact surface of touch sensor 1 (X1P)
5016	Distance to X-axis - contact surface of touch sensor 1 (X1M)
5017	Distance to Z-axis + contact surface of touch sensor 1 (Z1P)
5018	Distance to Z-axis - contact surface of touch sensor 1 (Z1M)
5056	Distance to X-axis + contact surface of touch sensor 2 (X2P)
5057	Distance to X-axis - contact surface of touch sensor 2 (X2M)
5058	Distance to Z-axis + contact surface of touch sensor 2 (Z2P)
5059	Distance to Z-axis - contact surface of touch sensor 2 (Z2M)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
This parameter is related to the function for direct input of offset value measured B.
Set the distance (signed) from a measurement reference position to each contact surface of a sensor. For a diameter specification axis, set a diameter value.
When arbitrary angular axis control is performed, set the distance in the Cartesian coordinate system.

> **NOTE**
> Parameters Nos. 5056 to 5059 are valid when bit 0 (2NR) of parameter No. 5051 is set to 1.





5020	**Tool offset number used with the function for direct input of offset value measured B**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to number of tool compensation values

Set a tool offset number used with the function for direct input of offset value measured B (when a workpiece coordinate system shift amount is set). (Set the tool offset number corresponding to a tool under measurement beforehand.) This parameter is valid when automatic tool offset number selection is not performed (when bit 5 (QNI) of parameter No. 5005 is set to 0).

5021	Number of interpolation cycles of pulses stored until the tool is about to touch the touch sensor

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 8

When a touch sensor with one contact signal input is used for the "direct input of offset value measured B" function or when the movement direction determination specification is enabled, set the number of interpolation cycles of pulses stored until the tool is about to touch the touch sensor by manual operation. When 0 is set, the specification of the maximum value 8 is assumed.

NOTE
This parameter is valid when bit 3 (TS1) of parameter No. 5004 or bit 4 (TSD) of parameter No. 5009 is set to 1.

5024	Number of tool compensation values

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 999

Set the maximum allowable number of tool compensation values used for each path.
Ensure that the total number of values set in parameter No. 5024 for the individual paths is within the number of compensation values usable in the entire system. The number of compensation values usable in the entire system depends on the option configuration.
If the total number of values set in parameter No. 5024 for the individual paths exceeds the number of compensation values usable in the entire system, or 0 is set in parameter No. 5024 for all paths, the number of compensation values usable for each path is a value obtained by dividing the number of compensation values usable in the entire system by the number of paths.
Tool compensation values as many as the number of compensation values used for each path are displayed on the screen. If tool compensation numbers more than the numbers of compensation values usable for each path are specified, alarm PS0115 “VARIABLE NO. OUT OF RANGE” is issued.
For example, 64 tool compensation sets are used, 20 sets may be allocated to path 1, 30 sets to path 2, and 14 sets to path 3. All of 64 sets need not be used.

	#7	#6	#5	#4	#3	#2	#1	#0
5051							2AT	2NR

[Input type] Parameter input
[Data type] Bit path

#0 2NR When the direct input of offset value measured B for 2 spindle lathe is used:
0: One touch sensor is used.
1: Two touch sensors are used.

#1 2AT When a workpiece coordinate system shift amount is set in the workpiece coordinate system memory with the direct input of offset value measured B for 2 spindle lathe:
0: A setting is made at the current cursor position.
1: An automatic selection is made.

5053	Tool compensation number shift amount for the direct input of offset value measured B for 2 spindle lathe

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to number of tool compensation values

When the direct input of offset value measured B for 2 spindle lathe is used, tool compensation numbers used to set measured tool compensation values are divided into two groups, one for spindle 1 and the other for spindle 2.

[Example] When there are 32 tool offset pairs

	Setting	
	8	10
Spindle 1	1 to 8	1 to 10
Spindle 2	9 to 32	11 to 32

When this parameter is set to 0 or a value greater than the maximum number of tool offset pairs, the table below is applied.

Number of tool offset pairs	32 pairs	64 pairs	99 pairs	200 pairs	400 pairs	499 pairs	999 pairs
Spindle 1	1 to 16	1 to 32	1 to 49	1 to 100	1 to 200	1 to 249	1 to 499
Spindle 2	17 to 32	33 to 64	50 to 98	101 to 200	201 to 400	250 to 498	500 to 998

5054	Workpiece coordinate system memory for spindle 1

5055	Workpiece coordinate system memory for spindle 2

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 54 to 59

Specify a workpiece coordinate system from G54 to G59 for which a workpiece coordinate system shift amount is set. When parameter No. 5054 or No. 5055 is set to 0 or a value beyond the valid data range, the specification of 54 is assumed for the workpiece coordinate system memory for spindle 1, or the specification of 57 is assumed for the workpiece coordinate system memory for spindle 2.

NOTE
These parameters are valid when bit 1 (2AT) of parameter No. 5051 is set to 1.

Alarm and message

Number	Message	Description
PS5195	DIRECTION CAN NOT BE JUDGED	Measurement is invalid in the tool compensation measurement value direct input B function. [For 1-contact input] 1. The recorded pulse direction is not constant. - The machine is at a stop in the offset write mode. - The servo power is off. - Pulse directions are diverse. 2. The tool is moving along the two axes (X-axis and Z-axis). [For the movement direction discrimination specification] 1. The recorded pulse direction is not constant. - The machine is at a stop in the offset write mode. - The servo power is off. - Pulse directions are diverse. 2. The tool is moving along the two axes (X-axis and Z-axis). 3. The direction indicated by the tool compensation write signal does not match the movement direction of the axis.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64694EN)	Setting and displaying data

15.4.3 Chattering Prevention of "Direct Input of Offset Value Measured B"

Overview

This function is provided by the "direct input of offset value measured B" function to prevent an invalid offset value from being set even when the input signal from a touch sensor chatters due to mechanical vibration.

Explanation

If the "direct input of offset value measured B" function encounters a chattering input signal from a touch sensor, for example, due to mechanical vibration, the same signal input later can be ignored. Once an offset value is measured with a signal from the touch sensor, any signal input that follows is rejected until movement over at least the distance set by parameter No. 5019 later is specified. Signals are ignored if input.

Assuming that an offset value is measured correctly at point A in Fig. 15.4.3 (a), even when movement from A to B is specified, measurement at point C is invalid and ignored as the travel distance is smaller than the setting of parameter No. 5019.

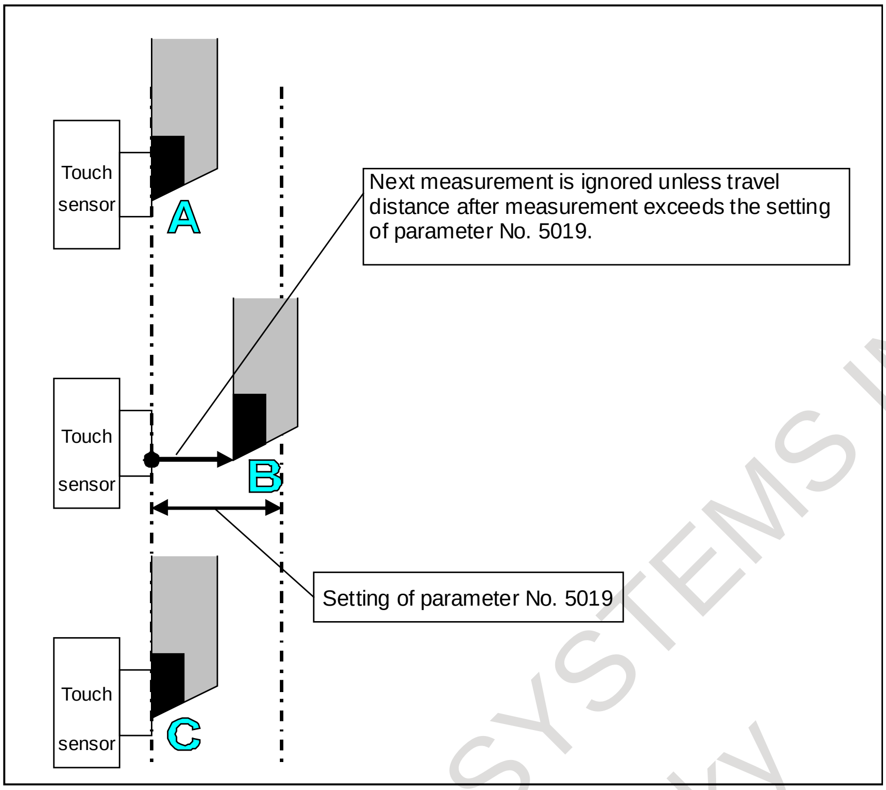


Fig. 15.4.3 (a) Chattering prevention 1

When movement from A to D is specified, measurement at point E is valid and an offset value is measured as the travel distance is larger than the setting of parameter No. 5019.

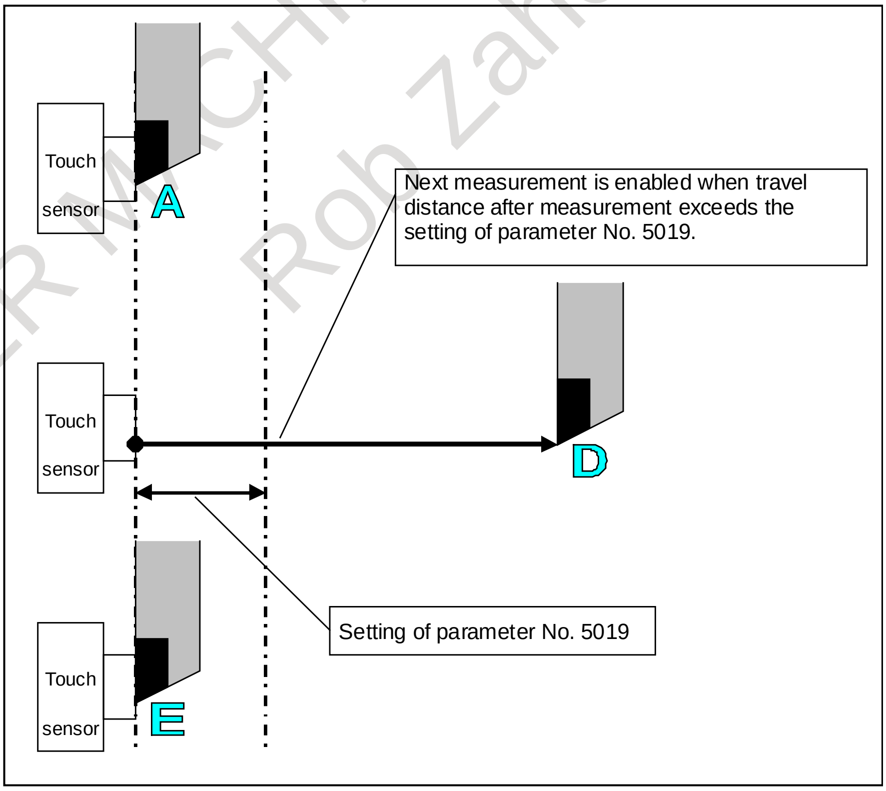


Fig. 15.4.3 (b) Chattering prevention 2

Parameter

5019	Chattering preventive distance of "direct input of offset value measured B"

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection
[Valid data range] 0 to 999999999

Once a signal is input from a touch sensor, any signal input from the touch sensor is ignored until movement over at least the distance set by this parameter is specified. If the setting is 0, this function is disabled and any input from the touch sensor is always accepted. Note that changing the setting of this parameter cancels the input signal ignoring status.

Caution

⚠ CAUTION

1. This function is available to both of tool offset value setting and workpiece coordinate shift amount setting.
2. The function is valid whether the contact detection type of the touch sensor is four-contact or one-contact.

16 PMC CONTROL FUNCTION

16.1 PMC AXIS CONTROL

16.1.1 PMC Axis Control

Overview

The PMC signals can directly control any given axis, independent of the CNC. An axis can be controlled by signals alone without using an NC program. For example, by specifying an amount of travel, feedrate, and so forth from the PMC, a movement can be made along an axis independently of other axes operated under CNC control. This enables the control of turrets, pallets, index tables and other peripheral devices using any given axes of the CNC.

Whether the CNC or PMC controls an axis is determined by the input signal provided for that particular axis.

Table 16.1.1 (a) Commands that can be executed by PMC axis control

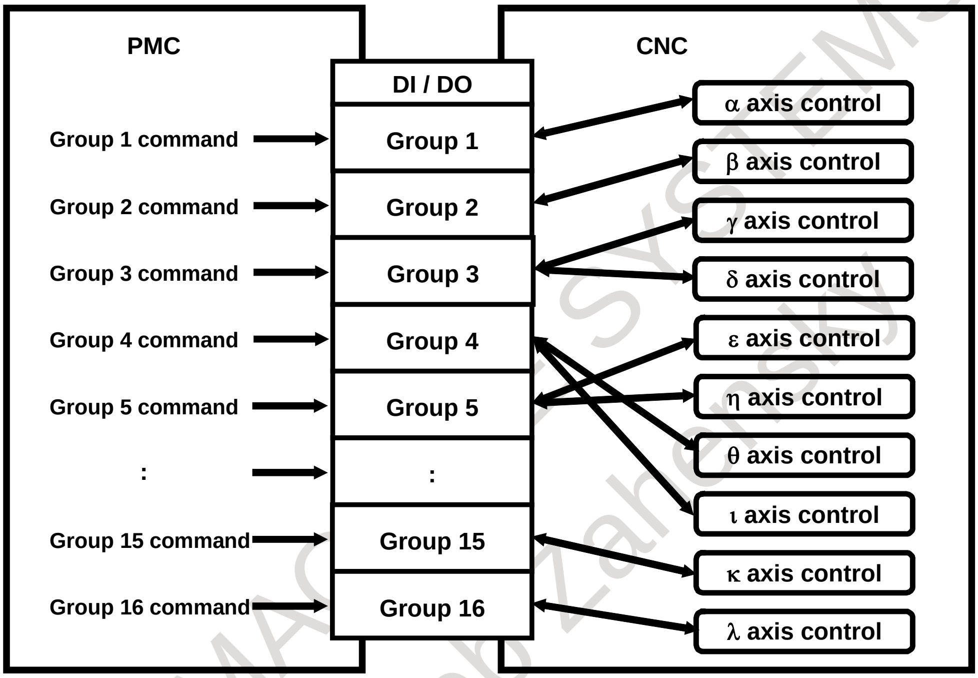
Rapid traverse
Cutting feed - feed per minute
Cutting feed - feed per revolution
Skip - feed per minute
Dwell
Reference position return
Continuous feed
1st to 4th reference position return
External pulse synchronization - Position code
External pulse synchronization - 1st to 3rd manual handle
Feedrate control command
Torque control command
Auxiliary function, Auxiliary function 2, Auxiliary function 3
Selection of the machine coordinate system
Cutting feed - sec/block specification

NOTE

Ladder sequence of the PMC axis control can be easily incorporated by using PMC function. Functional instruction AXCTL(Axis Control Command by PMC: SUB 53) that can easily realize the PMC axis control command is provided.
In the FANUC LADDER-III CD, function block that can be incorporated easily arious PMC axis control command is stored in the form of a library.
For details on functional instruction AXCTL and function block, refer to the PMC Programming Manual (B-64513EN). For details on library of function block, refer to the documents in the FANUC LADDER-III CD.

Explanation

Under PMC axis control, various types of control are exercised using signals.
To allow the command that can be executed by PMC axis control operations indicated in Table 16.1.1 (a) by PMC, maximum 16 groups of input/output signals are available. The groups are named group 1, group 2, ..., group 15, and group 16. (The number of available groups depends on the number of NC path.)
Each of these groups of input/output signals used with PMC axis control serves as the unit of PMC axis control. Which axis is to be controlled by each group must be set in parameter No. 8010 beforehand.(See Supplement 1 of Concept of PMC axis control Fig. 16.1.1 (a))
One group may be able to be assigned to multiple axes so that the same operation can be performed on the multiple axes. (Supplement 2)
By commanding multiple groups at the same time, multiple axes can be controlled independently. (Supplement 3)



(Supplement 1) The relationships between groups and axes are set in parameter No. 8010.
(Supplement 2) Multiple axes can be specified as with group 3.
(Supplement 3) Group 1 and group 2 can be specified independently.

Fig. 16.1.1 (a) Concept of PMC axis control

- Signals

Three types of signals are used with PMC control:
Group-by-group signals are assigned to each group.
Axis-by-axis signals are assigned to each axis.
Path-by-path signals are assigned to each path.

- Relationships between groups and axes (group-by-group signals)

Group-by-group signal addresses are allocated in the signal area of each path. (See Allocation of group-by-group signals (Table 16.1.1 (b)).)
Note, however, that PMC axis control and multi-path control are independent of each other.

NOTE
The number of available groups depends on the number of NC path or type of PMC.
In general, groups equal to 4 times the number of NC path are available.

Table 16.1.1 (b) Allocation of group-by-group signals

Group-by-group signal	Input signal (G) address	Output signal (F) address
Group 1	G0142 to G0149, G0150.5, G0151	F0130 to F0132, F0142
Group 2	G0154 to G0161, G0162.5, G0163	F0133 to F0135, F0145
Group 3	G0166 to G0173, G0174.5, G0175	F0136 to F0138, F0148
Group 4	G0178 to G0185, G0186.5, G0187	F0139 to F0141, F0151
Group 5	G1142 to G1149, G1150.5, G1151	F1130 to F1132, F1142
Group 6	G1154 to G1161, G1162.5, G1163	F1133 to F1135, F1145
Group 7	G1166 to G1173, G1174.5, G1175	F1136 to F1138, F1148
Group 8	G1178 to G1185, G1186.5, G1187	F1139 to F1141, F1151
Group 9	G2142 to G2149, G2150.5, G2151	F2130 to F2132, F2142
Group 10	G2154 to G2161, G2162.5, G2163	F2133 to F2135, F2145
Group 11	G2166 to G2173, G2174.5, G2175	F2136 to F2138, F2148
Group 12	G2178 to G2185, G2186.5, G2187	F2139 to F2141, F2151
Group 13	G3142 to G3149, G3150.5, G3151	F3130 to F3132, F3142
Group 14	G3154 to G3161, G3162.5, G3163	F3133 to F3135, F3145
Group 15	G3166 to G3173, G3174.5, G3175	F3136 to F3138, F3148
Group 16	G3178 to G3185, G3186.5, G3187	F3139 to F3141, F3151

The notation of group-by-group signals is described below.
For example, the group-by-group signals include the axis control command read signal EBUF (described later). This signal is represented as:
EBUFg#p
A group number is represented by "p" and "g".
"g" is replaced with A, B, C, or D, and A = 1, B = 2, C=3, and D = 4 are assumed.
"p" is replaced with a number from 1 to 4.
A group number is related with "p" and "g" as follows:
Group number = g+(p-1)×4

EBUFA#1 represents group 1 (1+(1-1)×4).
EBUFB#2 represents group 6 (2+(2-1)×4).
EBUFC#3 represents group 11 (3+(3-1)×4).
EBUFD#4 represents group 16 (4+(4-1)×4).

Note, however, for the small letter of “g” or “p”, that a signal named EBUFg#p does not actually exist.

In the text, signal names such as "EBUFA", "EBUFB", "EBUFC", "EBUFD", and "EBUFg" are used. Unless otherwise noted, these signal names are the abbreviations of "EBUFA#1" for group 1, "EBUFB#1" for group 2, "EBUFC#1" for group 3, "EBUFD#1" for group 4, and "EBUFg#1 to EBUFg#4" for general EBUF signals, respectively.

When assigning multiple axes to one group, set parameter No. 8010 so that the axes belong to the same path. A setting in which the assigned axes do not belong to the same path is erroneous (see "Example of incorrect relationships between groups and axes"). (Fig. 16.1.1 (c))
In Example of correct relationships between groups and axes (Fig. 16.1.1 (b)) , the first and second axes specified with group 1 belong to the same path (path 1), and the third and fourth axes specified with group 2 belong to the same path (path 2), so that operation is performed correctly.

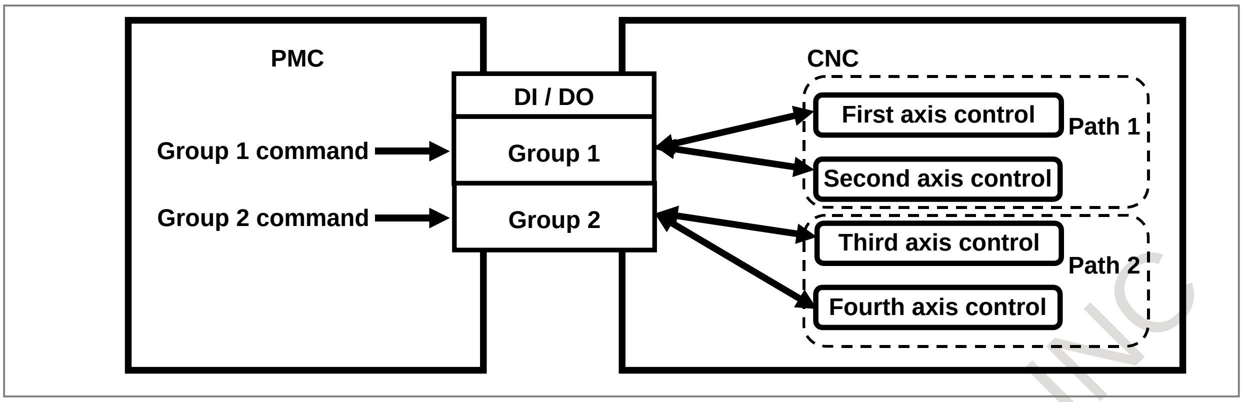


Fig. 16.1.1 (b) Example of correct relationships between groups and axes

In Fig. 16.1.1 (c), " Example of incorrect relationships between groups and axes", the first and second axes specified with group 1 belong to path 1, but the third axis specified with group 1 belongs to path 2, so that an intended operation is not performed.

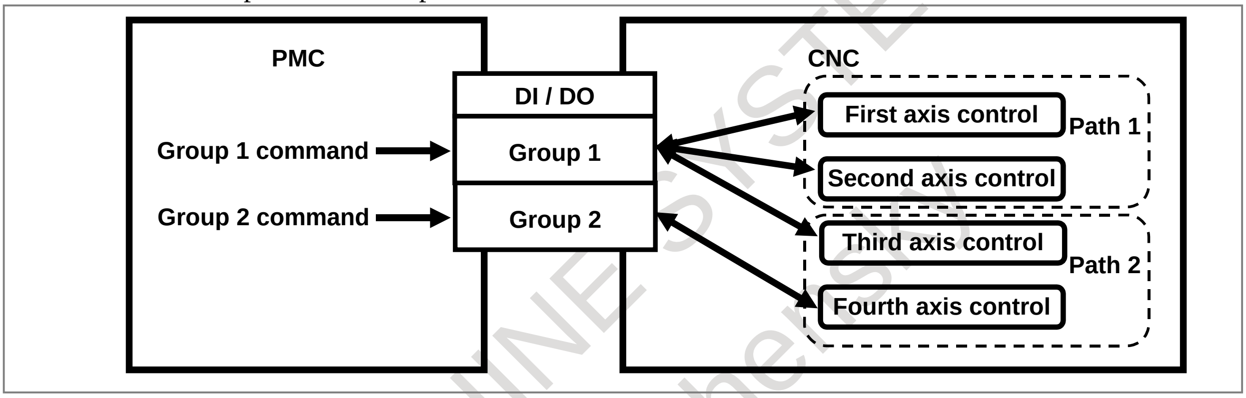


Fig. 16.1.1 (c) Example of incorrect relationships between groups and axes

As mentioned earlier, in PMC axis control, group-by-group signal addresses are allocated in the signal area of each path, but PMC axis control and multi-path control are independent of each other.
So, from the path to which group-by-group signal addresses are allocated, commands can be issued to axes that belong to another path. In this case, however, check that the target axes belong to the same path.

In Fig. 16.1.1 (d), “Commands to different paths”, the group-by-group signal addresses of group 1 are allocated to path 1, and the group-by-group signal addresses of group 5 are allocated to path 2. (See Table 16.1.1 (b), “Allocation of group-by-group signals”.)

Group 1 can issue commands to the axes that belong to path 2. Similarly, group 5 can issue commands to the axes that belong to path 1.

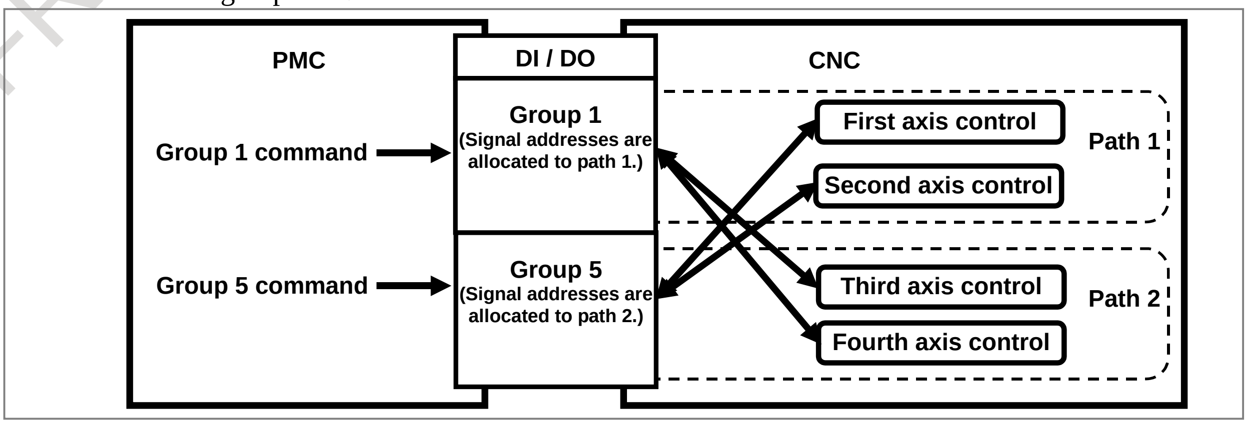


Fig. 16.1.1 (d) Commands to different paths

- Relationships between paths and groups (path-by-path signals, path-by-path parameters)

Path-by-path signal addresses are allocated in the signal area of each path. (See Table 16.1.1 (c), "Example of Path-by-path signal (PMC axis control override signal) allocation".)
However, PMC axis control and multi-path control are independent of each other.
The path-by-path direct signal (X) is described later.

> **NOTE**
> The number of available groups depends on the number of NC path or type of PMC.
> Generally, the maximum number of available groups equals the number of NC paths multiplied by 4.

Table 16.1.1 (c) Example of Path-by-path signal (PMC axis control override signal) allocation

Group-by-group signal	Input signal (G) address	Output signal (F) address
Group 1	G0150.0,1,6,7, G0151	F0129.5,7
Group 2	G0150.0,1,6,7, G0151	F0129.5,7
Group 3	G0150.0,1,6,7, G0151	F0129.5,7
Group 4	G0150.0,1,6,7, G0151	F0129.5,7
Group 5	G1150.0,1,6,7, G1151	F1129.5,7
Group 6	G1150.0,1,6,7, G1151	F1129.5,7
Group 7	G1150.0,1,6,7, G1151	F1129.5,7
Group 8	G1150.0,1,6,7, G1151	F1129.5,7
Group 9	G2150.0,1,6,7, G2151	F2129.5,7
Group 10	G2150.0,1,6,7, G2151	F2129.5,7
Group 11	G2150.0,1,6,7, G2151	F2129.5,7
Group 12	G2150.0,1,6,7, G2151	F2129.5,7
Group 13	G3150.0,1,6,7, G3151	F3129.5,7
Group 14	G3150.0,1,6,7, G3151	F3129.5,7
Group 15	G3150.0,1,6,7, G3151	F3129.5,7
Group 16	G3150.0,1,6,7, G3151	F3129.5,7

A path-by-path signal is input/output to and from a signal of the path to which the group belongs (for example, path 1 when a command is issued from group 1). No path-by-path signal is input/output to and from the path to which the axis assigned to the group belongs.

In Fig. 16.1.1 (e), "Relationships between paths and groups for path-by-path signals (G, F)", for example, the signal EBUFB#2 (described later) input from group 5 (whose signal addresses are allocated to path 2) is output to EOV#2 (path-by-path signal described later) of path 2 to which group 5 addresses are allocated.
EBUFB#2 is not output to EOV#1 of path 1, to which the first axis assigned to group 5 belongs.

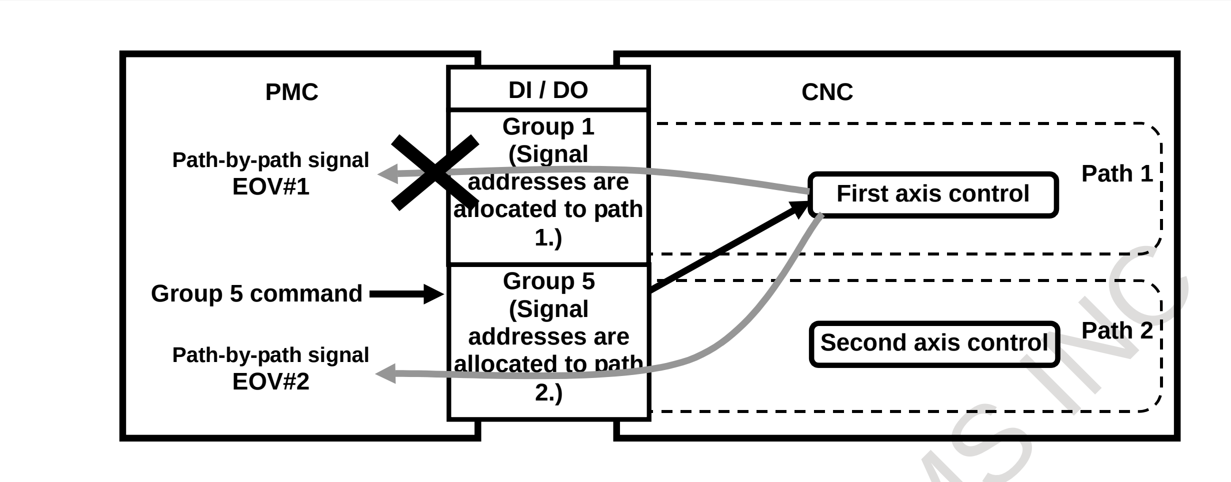


Fig. 16.1.1 (e) Relationships between paths and groups for path-by-path signals (G, F)

Restrictions on the number of groups

Basically, PMC axis control and multi-path control are independent of each other, but there are the following restrictions.

1. When PMC/L is used as PMC, the groups up to group 8 can be used. Group 9 and later groups cannot be used.

2. When PMC control is performed with more than four times as many groups as the number of NC paths, the signals shown in (A) and (B) below cannot be used for the excess paths and groups.
   To use feedrate override signals, etc., set bit 2 (OVE) of parameter No. 8001 to 1.
   (A) Path-by-path signals common to all groups
   Fx129.7 *EAXSL
   Fx129.5 EOV0
   (B) Signals same as the CNC
   Gx012 *FV0-*FV7
   Gx006.4 OVC
   Gx014.0,1 ROV1,2
   Gx046.7 DRN
   Gx019.7 RT

For example, when the number of NC paths is 2, the signals in (A) and (B) can be used by the groups up to group 8.

The restrictions on the number of NC paths, available (A) signals, and available (B) signals are shown Table 16.1.1 (d).

Table 16.1.1 (d)

Number of NC paths	Available (A) signals	Available (B) signals
1	F129.5,7	(B) signals in the range from G0 to G0767
2	F129.5,7,…, F1129.5,7	(B) signals in the range from G0 to G1767
3	F129.5,7,…, F2129.5,7	(B) signals in the range from G0 to G2767
4	F129.5,7,…, F3129.5,7	(B) signals in the range from G0 to G3767

- Direct signal, path-by-path parameter

The direct signal (X) and path-by-path parameter differ from the path-by-path signals (G, F).
The direct signals and parameter values set for the paths to which all axes related to a group belong are valid.
The direct signals and parameter values are irrelevant to values set for the path to which a group belongs.

In Fig. 16.1.1 (f), "Relationships between paths and groups for the direct signal (X)", when a command is issued from group 5 to an axis of path 1, the valid direct signal ESKIP (described later) is ESKIP#1 of path 1. The signal ESKIP#2 of path 2 is invalid.

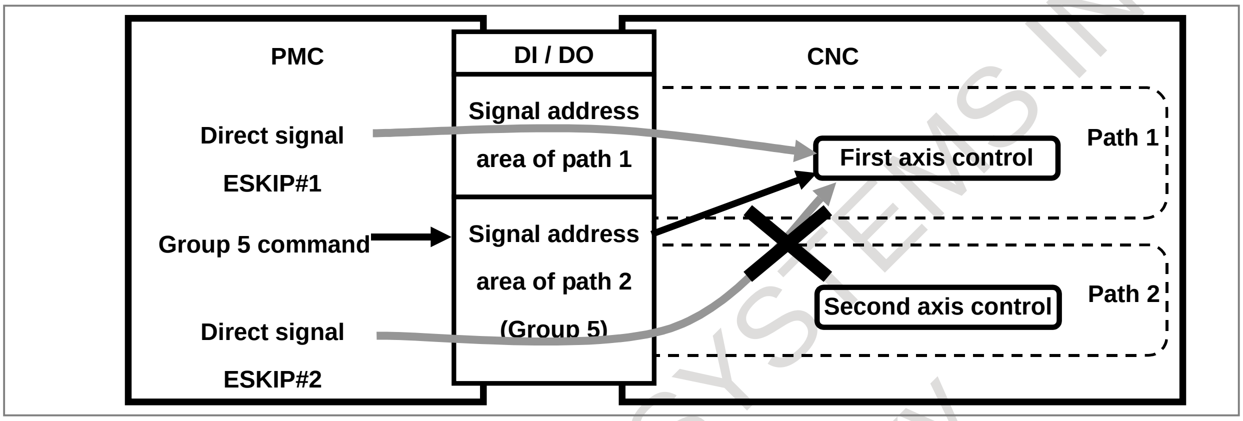


Fig. 16.1.1 (f) Relationships between paths and groups for the direct signal (X)

In Fig. 16.1.1 (g), "Relationships between path-by-path parameters and groups", when a command is issued from group 5 to path 1, the setting value of 0 with bit 2 (OVE) of the path-by-path parameter No. 8001 for path 1 is valid.

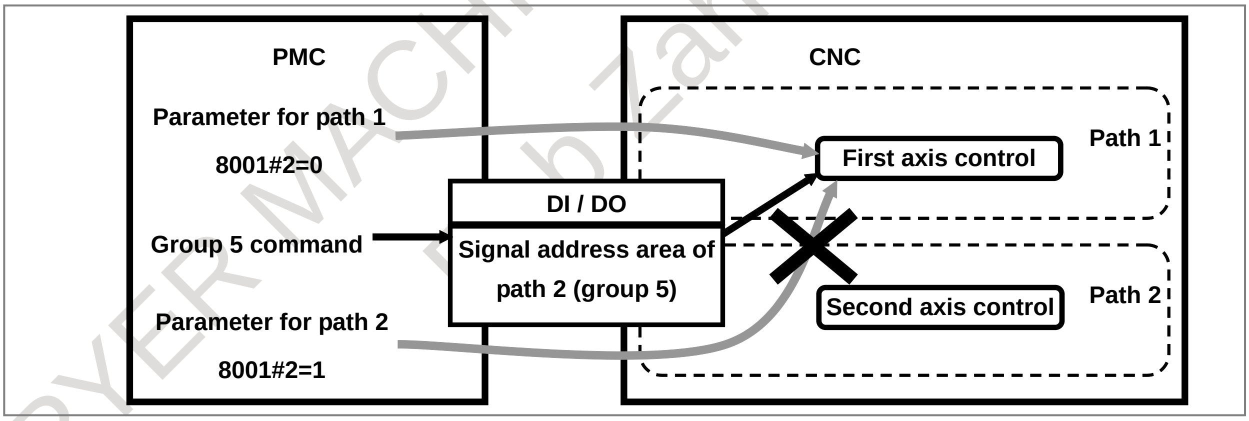


Fig. 16.1.1 (g) Relationships between path-by-path parameters and groups

- Relationships between axes and paths (axis-by-axis signals, axis-by-axis parameters)

The specifications of the axis-by-axis signals and axis-by-axis parameters associated with the relationships between paths and axes are the same as the NC specifications.
Signals and parameters set for an axis are valid for the axis, and are not affected by groups.

PMC axis control consists of groups, axes, and paths.

Among the three relationships, that is, the group-axis, path-group, and path-axis relationships, the two relationships other than the path-axis relationship represent a concept specific to PMC axis control.

⚠ **CAUTION**

1 PMC axis control cannot be used to an axis under the axis synchronous control. Cancel axis synchronous control before starting PMC axis command.
2 PMC axis control cannot be used to an axis under synchronous slave axis of synchronous/composite control. Cancel synchronous control before starting PMC axis command in slave axis.
3 Each axes cannot execute another PMC axis command by assigning two or more axes to the same group. Assign the axes to another groups, and command to each group.

Basic procedure

(1) In parameter No. 8010, set one of groups 1 to 16 for use with each axis to be used.
When movements are made along multiple axes for synchronization by using one group, ensure that the feedrate-related parameters (rapid traverse, acceleration/deceleration time constant, diameter/ radius specification, linear axis/rotation axis, and so forth) must be set to the same conditions.
When assigning multiple axes to one group, check that all axes that belong to the group are in the same path.

(2) To detach from CNC control and to enable direct PMC axis control, set each control axis selection signal (EAX1 to EAX8) bit, that corresponds to an axis to be controlled, to "1".

(3) Determine the operation.
The axis control command signals (EC0g to EC6g) specify the type of operation. The axis control feedrate signals (EIF0g to EIF15g) specify the feedrate. The axis control data signals (EID0g to EID31g) specify the moving distance and other data.
These signals, together with block stop prohibition signal EMSBKg (described later), determine one complete operation, which is tantamount to one block command executed during CNC-controlled automatic operation. These signals may be collectively called the axis control block data signals. (Refer to Table 16.1.1 (e), "List of signals determining data, tantamount to one block for PMC axis control".)

Table 16.1.1 (e) List of signals determining data, tantamount to one block for PMC axis control

Generic name	Signal name	Symbol	Data type
Axis control block data signals	Block stop prohibition signal	EMSBKg	Bit
	Axis control command signal	EC0g to EC6g	Byte
	Axis control feedrate signal	EIF0g to EIF15g	Word
	Axis control data signal	EID0g to EID31g	2-word

(4) When the data governing a complete operation (one block) is determined, reverse the logical state of the axis control command read signal EBUFg (i.e., from "0" to "1" or vice versa). Note that, for this to occur, axis control command read completion signal EBSYg must be in the same logical state as EBUFg.

(5) The CNC is capable of storing axis control functions from the PMC in its buffer so that multiple operations can be performed in series, under the control of the PMC. This allows the CNC to accept a new command block from the PMC during the execution of another block if the buffer has free space.
The Fig. 16.1.1 (h), "Buffering under PMC axis control" illustrates an example in which command [1] is being executed, commands [2] and [3] are stored in the buffers, and command [4] has been issued (the axis control block data signal is set).

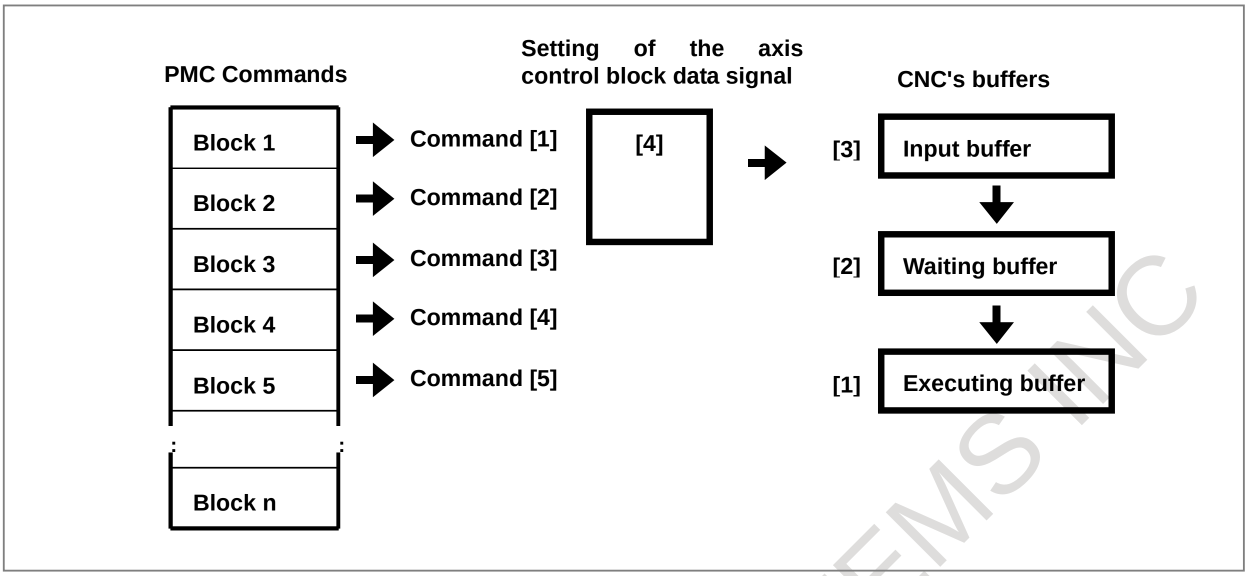


Fig. 16.1.1 (h) Buffering under PMC axis control

When the execution of command [1] is completed:
Command [2] is transferred from the waiting buffer to the executing buffer;
Command [3] is transferred from the input buffer to the waiting buffer;
and
Command [4] is transferred to the input buffer as the command block (axis control block data signal).
After the reception of command [4] by the input buffer, the PMC can issue command [5] to the CNC (the axis control block data signal is set).
The timing chart for the command operation is shown below. (Fig. 16.1.1 (i), "Timing chart of command operation")

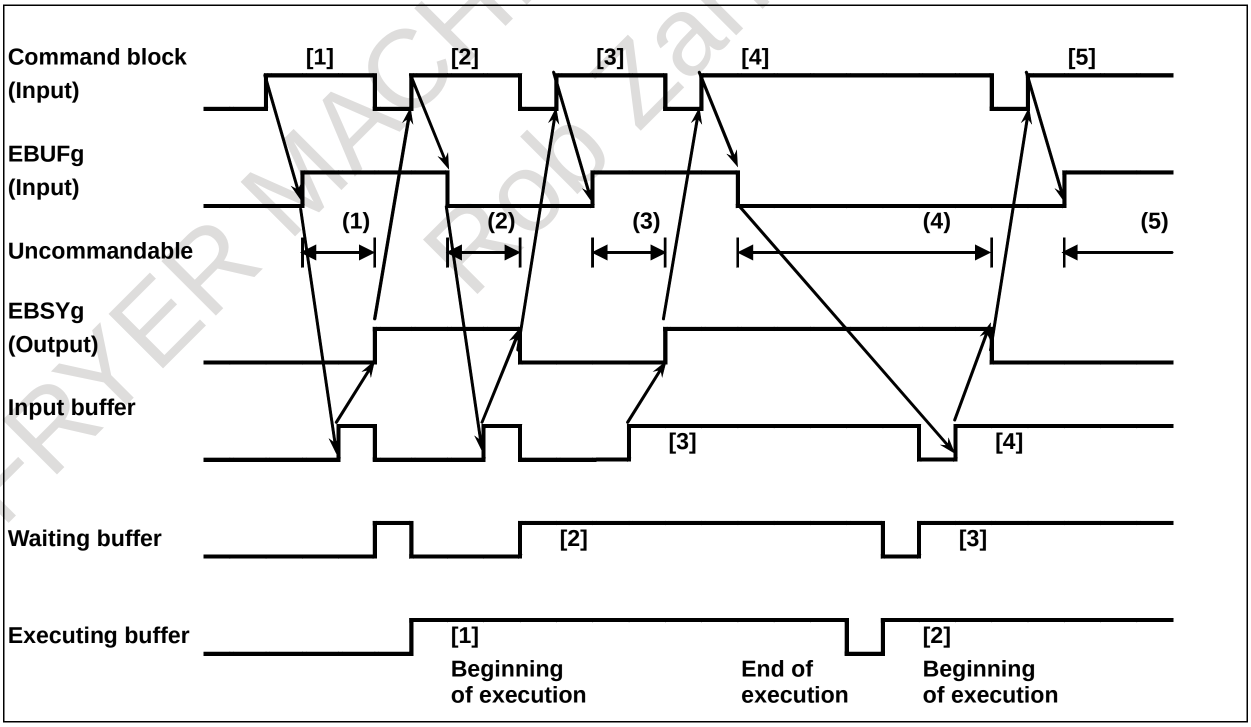


Fig. 16.1.1 (i) Timing chart of command operation

(1), (2), (3), (4), (5) : A new block cannot be issued during these intervals (while EBUFg and EBSYg are in different logical states). In interval (4), the buffer is in the "full" state.

The status of the CNC buffer can be determined by the exclusive OR of the axis control command read signal EBUFg, input from the PMC, and axis control command read completion signal EBSYg, output from the CNC. (Table 16.1.1 (f), "Buffering status in PMC axis control")

Table 16.1.1 (f) Buffering status in PMC axis control

EBUFg	EBSYg	Exclusive OR (XOR)	CNC buffer status
0	0	0	The previous block has already been read into the CNC buffer. The PMC can issue the next block.
1	1		
0	1	1	The previous block has not yet been read completely. It is just being read or buffer is in the "full" state, and waiting for the CNC buffer to become available. Do not issue the next block, nor reverse the logical state of EBUFg. Reversing the EBUFg state invalidates any block that has been already issued.
1	0		

(6) Repeat steps (3) and (4) until all the blocks have been issued.
When the final block has been issued and other command is unnecessary, set control axis selection signals EAX1 to EAX8 to “0“. Before setting these signals to “0“, however, check that the blocks stored in the CNC's input, waiting, and executing buffers have all been executed. Setting the signals to “0“ while a block is being executed, or while a block remains in any of these buffers, results in the issue of a P/S alarm. This alarm PS0139, “CANNOT CHANGE PMC CONTROL AXIS” suspends the current block execution and invalidates the blocks stored in the input and waiting buffers.
To ensure no block is being executed, or that there are no blocks remaining in the input or waiting buffer, check that control axis selection status signal *EAXSL is set to “0“.
For those axes that are always subject to PMC control without switching the management between CNC and PMC, such as those controlling turrets, pallets, and ATCs, ensure that the EAX1 to EAX8 signals are always set to “1“. There is no need to set these signals to “0“ after issuing commands from the PMC to the CNC. When all command blocks have been executed (there are no blocks remaining to be executed), the CNC automatically stops execution.

(7) When control axis selection signals EAX1 to EAX8 are set from “1“ to “0“, control is returned to the CNC.

Command

Overview of commands

As described in step (3) in the basic procedure, one PMC axis control command block is represented by the axis control block data signals.

PMC axis control enables the commands indicated in Table 16.1.1 (g), “Command list” to be specified.

In Table 16.1.1 (g), "Command list":
"Command" represents the axis control command signals (EC0g to EC6g).
"Data 1" represents the axis control feedrate signals (EIF0g to EIF15g).
"Data 2" represents the axis control data signals (EID0g to EID31g).

Table 16.1.1 (g) Command list

No.	Command	Operation	Data 1	Data 2	Description
(1)	00h	Rapid traverse	Rapid traverse rate (NOTE)	Total moving distance	The same operation as “G00” of the CNC is performed.
(2)	01h	Cutting feed - feed per minute	Cutting feedrate	Total moving distance	The same operation as “G94 G01” of the CNC is performed.
(3)	02h	Cutting feed - feed per revolution	Feedrate per revolution	Total moving distance	The same operation as “G95 G01” of the CNC is performed.
(4)	03h	Skip - feed per minute	Cutting feedrate	Total moving distance	The same operation as “G31 G01” of the CNC is performed.
(5)	04h	Dwell	---	Dwell time	The same operation as “G04” of the CNC is performed.
(6)	05h	Reference position return	---	---	After a rapid traverse movement is made in the reference position return direction set by bit 5 (ZMIx) of parameter No. 1006, the same operation as manual reference position return of the CNC is performed.
(7)	06h	Continuous feed	Continuous feedrate	Feed direction (EID31g only)	A continuous feed operation is performed along a controlled axis in a certain direction. The same operation as continuous feed in the JOG mode of the CNC is performed.
(8)	07h	First reference position return	Rapid traverse rate (NOTE)	---	The same operation as reference position return from an intermediate position done with “G28” of the CNC is performed.
(9)	08h	Second reference position return	Rapid traverse rate (NOTE)	---	The same operation as reference position return from an intermediate position done with “G30 P2” of the CNC is performed.
(10)	09h	Third reference position return	Rapid traverse rate (NOTE)	---	The same operation as reference position return from an intermediate position done with ”G30 P3” of the CNC is performed.
(11)	0Ah	Fourth reference position return	Rapid traverse rate (NOTE)	---	The same operation as reference position return from an intermediate position done with G30 P4” the CNC is performed.
(12)	0Bh	External pulse synchronization - position coder	Pulse weight	---	Synchronous operation with the position coder is performed.
(13)	0Dh	External pulse synchronization - first manual handle	Pulse weight	---	Synchronous operation with the first manual handle is performed.
(14)	0Eh	External pulse synchronization - second manual handle	Pulse weight	---	Synchronous operation with the second manual handle is performed.
(15)	0Fh	External pulse synchronization - third manual handle	Pulse weight	---	Synchronous operation with the third manual handle is performed.
(16)	10h	Speed command	Speed specified by speed command	---	Continuous feed operation based on a speed command is performed.
(17)	11h	Torque control	Maximum feedrate	Torque data	Continuous feed operation based on torque control is performed.
(18)	12h	Auxiliary function 1	---	Auxiliary function code	The same operation as an auxiliary function of the CNC is performed.

No.	Command	Operation	Data 1	Data 2	Description
(19)	14h	Auxiliary function 2	---	Auxiliary function code	The same operation as an auxiliary function of the CNC is performed.
(20)	15h	Auxiliary function 3	---	Auxiliary function code	The same operation as an auxiliary function of the CNC is performed.
(21)	20h	Machine coordinate system selection	Rapid traverse rate (NOTE)	Machine coordinate value	The same operation as “G53” of the CNC is performed.
(22)	21h	Cutting feed - sec/block specification	Cutting feed time	Total moving distance	Cutting feed is performed according to a specified period of time. In the sec/block specification, specify a period of time required for ending a block.

NOTE
The rapid traverse rate is valid only when bit 0 (RPD) of parameter No. 8002 is set to 1.

Details of commands

A detailed description of each command is provided below. The parenthesized number following each command title represents the value of the axis control command signals EC0g to EC6g.

(1) Rapid traverse (00h)

This command performs the same operation as "G00" of the CNC.

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Rapid traverse command (00h)
EIF0g to EIF15g	G144,145	Rapid traverse rate
EID0g to EID31g	G146 to 149	Total moving distance

Rapid traverse rate

With bit 0 (RPD) of parameter No. 8002, whether the same rapid traverse rate as set in the parameter No. 1420 of the CNC is used or the feedrate of the PMC axis interface set by EIF0g to EIF15g is used independently of the CNC can be chosen.

When bit 0 (RPD) of parameter No. 8002 is set to 1, set a rapid traverse rate with a binary code.

[Unit of data]

		Unit of data	Unit
		IS-A to IS-C	
Linear axis	Metric machine	1	mm/min
	Inch machine	0.1	inch/min
Rotation axis		1	deg/min

NOTE
When bit 0 (CMI) of parameter No. 11850 is 0, the rapid traverse rate set by EIF0g to EIF15g is always represented in millimeters.

[Valid data range] 1 to 65535

Total moving distance

Set an incremental travel amount in the input system unit of the axis with a binary code.

[Unit of data] Least input increment of the applied axis (Refer to the standard parameter setting table (A))

[Valid data range]

IS-A	IS-B to IS-C
-99999999 to 99999999 (8-digit)	-999999999 to 999999999 (9-digit)

For diameter specification based on bit 3 (DIAx) of parameter No. 1006, whether to specify a radius value or diameter value can be chosen using bit 1 (CDI) of parameter No. 8005.

When bit 0 (RPD) of parameter No. 8002 is set to 1 (to use the PMC axis interface for rapid traverse rate selection), the rapid traverse rate unit is 1 mm/min if bit 2 (R10) of parameter No. 8005 is set to 0; the rapid traverse rate unit is 10 mm/min if bit 2 (R10) of parameter No. 8005 is set to 1.

With bit 2 (OVE) of parameter No. 8001, select a dry run signal and manual rapid traverse selection signal to be used. In addition to these signals, with bit 2 (OVE) of parameter No.8001, select also a feed rate override signal, a override cancellation signal and rapid traverse override signal. The Table 16.1.1 (h) indicates the relationships between the bit and signals.

Table 16.1.1 (h)

Signals	Bit 2 (OVE) of parameter No.8001 = 0 (same signals as those used for the CNC)	Bit 2 (OVE) of parameter No. 8001 = 1 (signals specific to the PMC)
Feedrate override signals	*FV0 to *FV7<G012>	*EFOV0 to *EFOV7<G151>
Override cancellation signal	OVC<G006.4>	EOVC<G150.5>
Rapid traverse override signals	ROV1,2<G014.0,1>	EROV1,2<G150.0,1> or *EROV0 to *EROV7<G151>
Dry run signal	DRN<G046.7>	EDRN<G150.7>
Manual rapid traverse selection signal	RT<G019.7>	ERT<G150.6>

When the dry run signal DRN/EDRN is set to “1”, the relationship of feedrate is indicated as Table 16.1.1 (i)

Table 16.1.1 (i)

Manual rapid traverse selection signal (RTorERT)	When rapid traverse is commanded	
	RDE(No.8001#3)=0	**RDE(No.8001#3)=1**
0	Rapid traverse	Dry run feedrate ×*JV (NOTE)
1	Rapid traverse	Rapid traverse

> **NOTE**
> When bit 1 (JOV) of parameter No. 1402 is set to 1, the manual feedrate override signal *JV is not applied, but the dry run feedrate is just applied.

Related parameters

Bit 3 (DIAx) of parameter No. 1006
Bit 1 (JOV) of parameter No. 1402
Parameter No. 1420
Bit 2 (OVE) of parameter No. 8001
Bit 3 (RDE) of parameter No. 8001
Bit 0 (RPD) of parameter No. 8002
Bit 1 (CDI) of parameter No. 8005
Bit 2 (R10) of parameter No. 8005

Related signals

Manual rapid traverse selection signal RT <G019.7>
Dry run signal DRN <G046.7>
Manual rapid traverse selection signal (for PMC axis control) ERT <G150.6>
Dry run signal (for PMC axis control) EDRN <G150.7>

(2) Cutting feed - feed per minute (01h)

This command performs the same operation as “G94 G01” of the CNC.

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Cutting feed - feed minute (01h)
EIF0g to EIF15g	G144,145	Cutting feedrate
EID0g to EID31g	G146 to 149	Total moving distance

Cutting feed rate

Set a feedrate along the axis with a binary code.
A set feedrate can be increased by a factor of 10 with bit 3 (F10) of parameter No. 8002.

When bit 4 (EFD) of parameter No. 8006 is set to 1, the unit of feedrate data increases by a factor of 100. Moreover, the feedrate unit can be changed using a combination of bit 4 (PF1) of parameter No. 8002 and bit 5 (PF2) of parameter No. 8002.

[Unit of data]When the bit 3 (F10) of parameter No. 8002 is set to 0.

		Unit of data			Unit
		IS-A	IS-B	IS-C	
Linear axis	Metric input	10	1	0.1	mm/min
	Inch input	0.1	0.01	0.001	inch/min
Rotation axis		10	1	0.1	deg/min

When the bit 3 (F10) of parameter No. 8002 is set to 1.

		Unit of data			Unit
		IS-A	IS-B	IS-C	
Linear axis	Metric input	100	10	1	mm/min
	Inch input	1	0.1	0.01	inch/min
Rotation axis		100	10	1	deg/min

[Valid data range] 1 to 65535

⚠ **CAUTION**

1 When “0” is specified, the buffering state remains unchanged and no axis movement takes place. In such a case, perform a reset operation with the reset signal ECLRg.
2 Clamping to the cutting feedrate (parameter No. 1430) is disabled.

Total moving distance

The specifications of the rapid traverse command (EC0g to EC6g: 00h) are applicable.

With bit 2 (OVE) of parameter No. 8001, select a dry run signal and manual rapid traverse selection signal to be used. In addition to these signals, with bit 2 (OVE) of parameter No.8001, select also a feed rate override signal, a override cancellation signal and rapid traverse override signal. The Table 16.1.1 (j) below indicates the relationships between the bit and signals.

Table 16.1.1 (j)

Signals	Bit 2 (OVE) of parameter No.8001 = 0 (same signals as those used for the CNC)	Bit 2 (OVE) of parameter No. 8001 = 1 (signals specific to the PMC)
Feedrate override signals	*FV0 to *FV7<G012>	*EFOV0 to *EFOV7<G151>
Override cancellation signal	OVC<G006.4)	EOVC<G150.5>
Rapid traverse override signals	ROV1,2<G014.0,1>	EROV1,2<G150.0,1> or *EROV0 to *EROV7<G151>
Dry run signal	DRN<G046.7>	EDRN<G150.7>
Manual rapid traverse selection signal	RT<G019.7>	ERT<G150.6>

When the dry run signal DRN/EDRN is set to “1”, the feedrate is as indicated Table 16.1.1 (k).
Unlike rapid traverse, the feedrate does not depend on the value of bit 3 (RDE) of parameter No. 8001.

Table 16.1.1 (k)

Manual rapid traverse selection signal (RT / ERT)	When cutting feed is specified
0	Dry run speed × *JV (Note)
1	Cutting feed

> **NOTE**
> When bit 1 (JOV) of parameter No. 1402 is set to 1, the manual feedrate override signal *JV is not applied, but the dry run feedrate is just applied.

Related parameters

Bit 3 (DIAx) of parameter No. 1006
Bit 1 (JOV) of parameter No. 1402
Bit 2 (OVE) of parameter No. 8001
Bit 3 (F10) of parameter No. 8002
Bit 4 (PF1) of parameter No. 8002
Bit 5 (PF2) of parameter No. 8002
Bit 1 (CDI) of parameter No. 8005
Bit 4 (EFD) of parameter No. 8006

Related signals

Manual rapid traverse selection signal RT <G019.7>
Dry run signal DRN<G046.7>
Manual rapid traverse selection signal (for PMC axis control) ERT <G150.6>
Dry run signal (for PMC axis control) EDRN <G150.7>

(3) Cutting feed - feed per revolution (02h)

This command performs the same operation as "G95 G01" of the CNC.
Set the amount of feed along the axis per spindle revolution.
The feedrate per spindle revolution depends on whether the M series or T series is used.

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Cutting feed - feed per revolution command (02h)
EIF0g to EIF15g	G144,145	Feedrate per revolution
EID0g to EID31g	G146 to 149	Total moving distance

Feedrate per revolution

<For the T series>

[Unit of data] The Table 16.1.1 (l) indicates the unit of data depending on the setting of bits 6 (FR1) and bit 7 (FR2) of parameter No. 8002.

Table 16.1.1 (l)

Parameter		Metric input (mm/rev)	Inch input (inch/rev)	Rotation axis (deg/rev)
FR2	**FR1**			
1	**1**	0.0001	0.000001	0.0001
0	**0**			
0	**1**	0.001	0.00001	0.001
1	**0**	0.01	0.0001	0.01

[Valid data range] 1 to 65535
(However, data within the ranges indicated Table 16.1.1 (m) must be commanded.)

Table 16.1.1 (m)

		Valid data range	Unit
		IS-A to IS-C	
Linear axis	**Metric input**	0.0001 to 500.0000	mm/rev
	Inch input	0.000001 to 9.999999	inch/rev
Rotation axis		0.0001 to 500.0000	deg/rev

<For M series>

[Unit of data] The Table 16.1.1 (n) indicates the unit of data depending on the setting of bits 6 (FR1) and bit 7 (FR2) of parameter No. 8002.

Table 16.1.1 (n)

Parameter		Metric input (mm/rev)	Inch input (inch/rev)	Rotation axis (deg/rev)
FR2	**FR1**			
1	**1**	0.01	0.0001	0.01
0	**0**			
0	**1**	0.1	0.001	0.1
1	**0**	1	0.01	1

[Valid data range] 1 to 65535
(However, data within the ranges indicated Table 16.1.1 (o) must be commanded.)

Table 16.1.1 (o)

		Valid data range	Unit
		IS-A to IS-C	
Linear axis	**Metric input**	0.01 to 500.00	mm/rev
	Inch input	0.0001 to 9.9999	inch/rev
Rotation axis		0.01 to 500.00	deg/rev

⚠ CAUTION

1. A set feedrate can be increased by a factor of 10 or 100 by setting bit 6 (FR1) and bit 7 (FR2) of parameter No. 8002.
2. The feedrate is clamped to a value not exceeding the setting of parameter No. 8022.
3. Override is enabled. Dry run depends on the value of bit 3 (DRR) of parameter No. 8005.

Total moving distance
The specifications of the rapid traverse command (EC0g to EC6g: 00h) are applicable.

Dry run operation follows the specifications of cutting feed - feed per minute.

Related parameters
Bit 3 (DRR) of parameter No. 8005
Parameter No. 8022
Others follow the specifications of the cutting feed - feed per minute.

Related signals
Same as for cutting feed - feed per minute

(4) Skip - feed per minute (03h)

Skip

This command performs the same operation as "G31 G01" of the CNC.
High-speed skip is disabled.

Available skip signal
Skip signal of PMC axis control uses the same skip signal SKIP<X0004.7> as CNC. When bit 7 (SKE) of parameter No.8001 is set to 1, unique skip signal ESKIP<X0004.6> of PMC axis control can be used. Also, when bit 2 (XSG) of parameter No.3008 is set to 1, arbitrary X address specified by the parameters Nos.3012, 3019 can be used as a skip signal. Available skip signal is shown in Table 16.1.1 (p).

Table 16.1.1 (p) Skip signal to be used in PMC axis control skip

Bit 7(SKE) of parameter No.8001	Bit 2 (XSG) of parameter No.3008	Available skip signal
0	0	SKIP <X0004.7>, SKIP[#2] <X0013.7>, SKIP[#3]<X0011.7>, path 4 can not use skip signal.
1	0	ESKIP <X0004.6>, ESKIP[#2] <X0013.6>, ESKIP[#3]<X0011.6>, path 4 can not use skip signal.
0	1	Bit 7 of X address set to the parameter No.3012
1	1	Bit 6 of X address set to the parameter No.3019

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Skip command (03h)
EIF0g to EIF15g	G144,145	Cutting feedrate
EID0g to EID31g	G146 to 149	Total moving distance

Cutting feedrate
The specifications of the cutting feed command (EC0g to EC6g: 01h) are applicable.

> ⚠ **CAUTION**
> Feedrate override, dry run, and automatic acceleration/deceleration are disabled.

Total moving distance
The specifications of the rapid traverse command (EC0g to EC6g: 00h) are applicable.

Acceleration/deceleration after interpolation

When bit 5 (PSA) of parameter No.8007 is set to 1, acceleration/deceleration after interpolation is enabled for skip command of PMC axis control. Type of acceleration/deceleration is set to bit 3 (ASL) and bit 4 (ASB) of parameter No. 6210, and time constant is set to parameter No.6280. As a result, different from acceleration/deceleration after interpolation of cutting feed of normal, acceleration/deceleration after interpolation for the skip function is available. Also, in case of type of exponential acceleration/deceleration, acceleration/deceleration after interpolation in cutting feed or continuous feed under PMC axis control is available by setting the time constant to parameter No.8030 and setting FL rate to parameter No.8031. Setting of acceleration/deceleration is shown in Table 16.1.1 (q). In case of CNC skip command (G31), when bit 7 (SKF) of parameter No.6200 is set to 1, acceleration/deceleration after interpolation of cutting feed is enabled. Setting of acceleration/deceleration is shown in Table 16.1.1 (r).

This function is enabled bit 1 (SEB) of parameter No.6201. When parameter SEB is set to 1, Skip position of system variables (#5061 to #5080, #100151 to #100182) in which the accumulated pulses and positional deviation due to acceleration/deceleration was considered and compensated can be read.

Table 16.1.1 (q) Setting of acceleration/deceleration after interpolation of PMC axis control skip, when bit 5 (PSA) of parameter No.8007 = 1.

Acceleration/deceleration setting		Bit 1 (CTBx) of parameter No.1610	Bit 0 (CTLx) of parameter No.1610	Bit 4 (ASB) of parameter No.6210	Bit 3 (ASL) of parameter No.6210	Time constant, FL rate parameter
Exponential acceleration/ deceleration	Same time constant and FL rate as cutting feed	0	0	0	0	No.1622 No.1623
	Time constant and FL rate different from cutting feed	0	0	0	0	No.8030 No.8031
Linear acceleration/ deceleration	Same time constant as cutting feed	0	1	0	0	No.1622
	Time constant different from cutting feed	0 / 1	0 / 1	0	1	No.6280
Bell-shaped acceleration/ deceleration	Same time constant as cutting feed	1	0	0	0	No.1622
	Time constant different from cutting feed	0 / 1	0 / 1	1	0	No.6280

Table 16.1.1 (r) Setting of acceleration/deceleration after interpolation of CNC skip command, when bit 7 (SKF) of parameter No.6200 = 1.

Acceleration/deceleration setting		Bit 1 (CTBx) of parameter No.1610	Bit 0 (CTLx) of parameter No.1610	Bit 4 (ASB) of parameter No.6210	Bit 3 (ASL) of parameter No.6210	Time constant, FL rate parameter
Exponential acceleration/ deceleration	Same time constant and FL rate as cutting feed	0	0	0	0	No.1622 No.1623
	Time constant and FL rate different from cutting feed	This setting does not exist.				
Linear acceleration/ deceleration	Same time constant as cutting feed	0	1	0	0	No.1622
	Time constant different from cutting feed	0 / 1	0 / 1	0	1	No.6280
Bell-shaped acceleration/ deceleration	Same time constant as cutting feed	1	0	0	0	No.1622
	Time constant different from cutting feed	0 / 1	0 / 1	1	0	No.6280

NOTE

1. Setting value of parameters Nos.8030, 8031 are the same as cutting feed under PMC axis control.
2. Setting value of bit 3 (ASL) and bit 4 (ASB) of parameter No.6210, parameter No.6280 are the same as CNC skip command (G31).
3. Multi-step skip function, high-speed skip function, torque limit skip function, skip function for EGB Axis, and dwell skip function can not be used in PMC axis control skip.

Related parameters

Parameter No. 3012
Parameter No. 3019
Bit 1 (SK0) of parameter No. 6200
When Bit 5 (PSA) of parameter No. 8007 is set to 1, Bit 1 (SEB) of parameter No. 6201
When Bit 5 (PSA) of parameter No. 8007 is set to 1, Bit 3 (ASL) of parameter No. 6210 and Bit 4 (ASB) of parameter No. 6210, No. 6280

Related signals

Skip signal SKIP<X0004.7>, SKIP[#2]<X0013.7>, SKIP[#3]<X0011.7>, ESKIP<X0004.6>, ESKIP[#2]<X0013.6>, ESKIP[#3]<X0011.6>

(5) Dwell (04h)

This command performs the same operation as "G04" of the CNC.

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Dwell command (04h)
EID0g to EID31g	G146 to 149	Dwell time

Dwell time

Set a dwell time with a binary code.

Valid data range	Unit
1 to 9999999	ms

When diameter specification is selected with bit 3 (DIAx) of parameter No. 1006, whether to specify a radius value or diameter value for dwell operation can be specified using bit 1 (CDI) of parameter No. 8005.

When the increment system is IS-C, the dwell time can be set to a least input increment of 0.1 msec with bit 1 (DWE) of parameter No. 8002.

Related parameters

Bit 3 (DIAx) of parameter No. 1006
Bit 1 (DWE) of parameter No. 8002
Bit 1 (CDI) of parameter No. 8005

(6) Reference position return (05h)

This command performs the same operation as manual reference position return of the CNC after rapid traverse in the reference position return direction set by bit 5 (ZMIx) of parameter No. 1006.

Reference position return without DOG

If bit 1 (DLZx) of parameter No. 1005 for the function for return the reference position without dogs is set for each axis, and reference position return operation is not performed even once after the power is turned on, a reference position return operation (positioning at a grid closest to the current position) can be performed without using the reference position return deceleration signal when a movement (positioning at a location near the reference position) is made in the direction set for each axis with the continuous feed command (EC0g to EC6g: 06h) and the reference position return command (EC0g to EC6g: 05h) is specified.
At the time of positioning at a location near the reference position, the machine needs to be moved in the reference position return direction at such a feedrate that the servo position deviation exceeds the value set in parameter No. 1836.
The direction from a neighborhood point to a grid is set by bit 5 (ZMIx) of parameter No. 1006.
If the reference position return command (EC0g to EC6g: 05h) is issued after reference position establishment, a high-speed reference position return operation is performed, regardless of the reference position return direction set by bit 5 (ZMIx) of parameter No. 1006.
Specify this command after canceling tool radius compensation and tool length compensation.

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Reference position return command (05h)

Related parameters

Bit 1 (DLZx) of parameter No. 1005
Bit 5 (ZMIx) of parameter No. 1006
Parameter No. 1836

(7) Continuous feed (06h)

This command performs a continuous feed operation along a controlled axis in a certain direction.
This command performs the same operation as continuous feed in the JOG mode of the CNC.

Continuous feed is performed until a reset. At this time, the remaining amount of travel is 0 at all times.

By setting the reset signal ECLRg to “1”, the command can be ended. At this time, the servo motor is decelerated to a stop, the axis moving signal EGENg is set to “0”, and the control axis selection status signal *EAXSL is also set to “0“.
Before specifying the next command, check that the control axis selection status signal *EAXSL is set to “0”.
Until the control axis selection status signal *EAXSL is set to “0”, ensure that the reset signal ECLRg continues to be set to “1”.

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Continuous feed command (06h)
EIF0g to EIF15g	G144,145	Continuous feedrate
EID31g	G149.7	Feed direction (specified with EID31g only)

Continuous feedrate
A feedrate along an axis is set in the same way as for cutting feed - feed per minute (EC0g to EC6g: 01h). A feedrate change can be made during continuous feed.
If a feedrate (EIF0g to EIF15g) is set, and the logical state of the axis control command read signal EBUFg is reversed, a new continuous feedrate is set.
This command is not buffered. So, the axis control command read completion signal EBSYg need not be checked when this command is specified usually.
A set feedrate can be increased by a factor of 10 with bit 3 (F10) of parameter No. 8002. Moreover, a set feedrate can be increased by a factor of up to 200 with bit 2 (JFM) of parameter No. 8004.

⚠ **CAUTION**
A maximum feedrate applicable when override is applied differs from a maximum feedrate applicable when override is canceled. Maximum feedrate data in the two cases is indicated Table 16.1.1 (s) and Table 16.1.1 (t).

Table 16.1.1 (s) Maximum feedrate (when an override of 254% is applied)

	IS-B		IS-C	
	Metric input (mm/min)	Inch input (inch/min)	Metric input (mm/min)	Inch input (inch/min)
1 times	166458	1664.58	16645	166.45
10 times	1664589	16645.89	166458	1664.58
200 times (NOTE 1)	19660500	196605.00	1966050	19660.50

Table 16.1.1 (t) Maximum feedrate (when override is canceled)

	IS-B		IS-C	
	Metric input (mm/min)	Inch input (inch/min)	Metric input (mm/min)	Inch input (inch/min)
1 times	65535	655.35	6553	65.53
10 times	655350	6553.5	65535	655.35
200 times (NOTE 1)	13107000	131070	1310700	13107

NOTE
1 A feedrate increased by a factor of 200 is valid only when the continuous feed command (EC0g to EC6g: 06h) is used.
2 The actual feedrate may not be displayed correctly, depending on the feedrate.

Feed direction
Specify the direction of continuous feed with the signal EID31g.
0: Positive direction
1: Negative direction
The signals EID0g to EID30g are undefined.
To change the feed direction during continuous feed, the signal EID31g is reversed and then the logical state of the axis control command read signal EBUFg is reversed.

Dry run operation follows the specifications of cutting feed - feed per minute.

Related parameters
Bit 1 (JOV) of parameter No. 1402
Bit 2 (OVE) of parameter No. 8001
Bit 3 (F10) of parameter No. 8002
Bit 2 (JFM) of parameter No. 8004

Related signals
Same as for cutting feed - feed per minute

(8) First reference position return (07h)

This command performs the same operation as reference position return from an intermediate position done with "G28" of the CNC.

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	First reference position return command (07h)
EIF0g to EIF15g	G144,145	Rapid traverse rate

Rapid traverse rate

The specifications of the rapid traverse command (EC0g to EC6g: 00h) are applicable.
When the first reference position return command is specified, the manual rapid traverse rate set by parameter No. 1424 is used if manual reference position return operation is not performed even once after the power is turned on.

> **NOTE**
> If bit 1 (DLZ) of parameter No. 1005 is set to enable the function for returning the reference position without dogs for each axis, and reference position return operation is not performed even once after the power is turned on, the alarm PS0090, “REFERENCE RETURN INCOMPLETE” is issued when the first reference position return command (EC0g to EC6g: 07h) is specified.

Dry run operation follows the specifications of rapid traverse.

Related parameters

Bit 1 (DLZx) of parameter No. 1005
Parameter No. 1240
Bit 1 (JOV) of parameter No. 1402
Parameter No. 1420
Parameter No. 1424
Bit 2 (OVE) of parameter No. 8001
Bit 3 (RDE) of parameter No. 8001
Bit 0 (RPD) of parameter No. 8002

Related signals

Same specification for rapid traverse command

(9) Second reference position return (08h)

This command performs the same operation as reference position return from an intermediate position done with "G30 P2" of the CNC.

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Second reference position return command (08h)
EIF0g to EIF15g	G144,145	Rapid traverse rate

Rapid traverse rate

The specifications of the rapid traverse command (EC0g to EC6g: 00h) are applicable.

Related parameters

Bit 1 (DLZx) of parameter No. 1005
Parameter No. 1241
Bit 1 (JOV) of parameter No. 1402
Parameter No. 1420
Bit 2 (OVE) of parameter No. 8001
Bit 3 (RDE) of parameter No. 8001
Bit 0 (RPD) of parameter No. 8002

Related signals

Same specification for rapid traverse command

(10) Third reference position return (09h)

This command performs the same operation as reference position return from an intermediate position done with "G30 P3" of the CNC.

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Third reference position return command (09h)
EIF0g to EIF15g	G144,145	Rapid traverse rate

Rapid traverse rate

The specifications of the rapid traverse command (EC0g to EC6g: 00h) are applicable.

Related parameters

Bit 1 (DLZx) of parameter No. 1005
Parameter No. 1242
Bit 1 (JOV) of parameter No. 1402
Parameter No. 1420
Bit 2 (OVE) of parameter No. 8001
Bit 3 (RDE) of parameter No. 8001
Bit 0 (RPD) of parameter No. 8002

Related signals

Same specification for rapid traverse command

(11) Fourth reference position return (0Ah)

This command performs the same operation as reference position return from an intermediate position done with "G30 P4" of the CNC.

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Fourth reference position return command (0Ah)
EIF0g to EIF15g	G144,145	Rapid traverse rate

Rapid traverse rate

The specifications of the rapid traverse command (EC0g to EC6g: 00h) are applicable.

Related parameters

Bit 1 (DLZx) of parameter No. 1005
Parameter No. 1243
Bit 1 (JOV) of parameter No. 1402
Parameter No. 1420
Bit 2 (OVE) of parameter No. 8001
Bit 3 (RDE) of parameter No. 8001
Bit 0 (RPD) of parameter No. 8002

Related signals

Same specification for rapid traverse command

(12) External pulse synchronization – position coder (0Bh)

This command performs a synchronous operation with the position coder. Using position coder of the serial spindle is also available.
When a negative external pulse value is specified, a movement is made in the opposite direction.
The remaining amount of travel is 0 at all times.
If a manual handle interrupt operation is performed for the same axis, an amount of travel based on external pulses added to manual handle interrupt pulses results.

By setting the reset signal ECLRg to "1", the command can be ended. At this time, the servo motor is decelerated to a stop, the axis moving signal EGENg is set to "0", and the control axis selection status signal *EAXSL is also set to "0".
Before specifying the next command, check that the control axis selection status signal *EAXSL is set to "0".
Until the control axis selection status signal *EAXSL is set to "0", ensure that the reset signal ECLRg continues to be set to "1".

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	External pulse synchronization - position coder command (0Bh)
EIF0g to EIF15g	G144,145	Pulse weight
EID0g to EID7g	G146	Spindle number of the serial spindle to be synchronized (common to the system) (*1)

*1: This data is set only during synchronization with the position coder of the serial spindle.

The type of position coder to be synchronized depends on the parameter setting, as shown Table 16.1.1 (u).

Table 16.1.1 (u)

Bit 3 (ESY) of parameter No. 8007	Bit 0 (EOS) of parameter No. 8019	Command of G146	Valid position coder
0	0/1	Not required	Position coder (not a serial spindle)
1	0	Not required	Position coder of the first spindle belonging to path 1 (*2)
1	1	Required	Position coder of an arbitrary spindle (set by G146)

*2: This can be commanded for the servo axis belonging to path 1.

Pulse weight

Set an external pulse weight.

1) In the Case of Manual pulse magnification change signal HNDMP<Gn088.3> is "0"
When setting a weight value, use the higher bits of signal (EIF8g to EIF15g) to specify the integer part of an external pulse weight, and use the lower bits of signal (EIF0g to EIF7g) to specify the fractional part.
The valid range is ±1/256 to ±127.
With EIF0g to EIF7g for the fractional part, specify a weight in steps of 1/256.

Example)
For 0.5: EIF0g to EIF7g = 80h (=80h/100h=128 / 256 = 0.5)
For 0.2: EIF0g to EIF7g = 33h (=33h/100h= 51 / 256 ≒ 0.2)

2) In the Case of Manual pulse magnification change signal HNDMP<Gn088.3> is “1”
When setting a weight value, use the higher bits of signal (EIF5g to EIF15g) to specify the integer part of an external pulse weight, and use the lower bits of signal (EIF0g to EIF4g) to specify the fractional part.
The valid range is ±1/32 to ±1023.
With EIF0g to EIF4g for the fractional part, specify a weight in steps of 1/32.

Example)
For 0.5: EIF0g to EIF4g = 10h (=10h/20h=16 / 32 = 0.5)
For 0.2: EIF0g to EIF4g = 06h (=06h/20h= 6 / 32 ≒ 0.2)

When a negative value is set to pulse weight, a movement is made along the axis in the opposite direction.
If a new pulse weight is set and the logical state of the axis control command read signal EBUFg is reversed during axis movement based on external pulse synchronization, a movement is made along the axis according to the new pulse weight.
This command is not buffered. So, the axis control command read completion signal EBSYg need not be checked when this command is specified normally.

Spindle number of a serial spindle to be synchronized (for synchronization with the position coder of a serial spindle)
The serial spindle to be synchronized is set with a spindle number common to the system.
The valid range extends from 1 to the maximum number of control spindle axes. If the spindle number falls outside the range, external pulse synchronization is disabled.

Example)
When the spindle (serial spindle) of each path is assigned as shown Table 16.1.1 (v) and Fig. 16.1.1 (j)

Table 16.1.1 (v)

Spindle number common to the system	Path number + Spindle number in the path
First spindle (S1)	First spindle in path 1 (S11)
Second spindle (S2)	Second spindle in path 1 (S12)
Third spindle (S3)	Third spindle in path 1 (S13)
Fourth spindle (S4)	First spindle in path 2 (S21)
Fifth spindle (S5)	First spindle in path 3 (S31)
Sixth spindle (S6)	Second spindle in path 3 (S32)

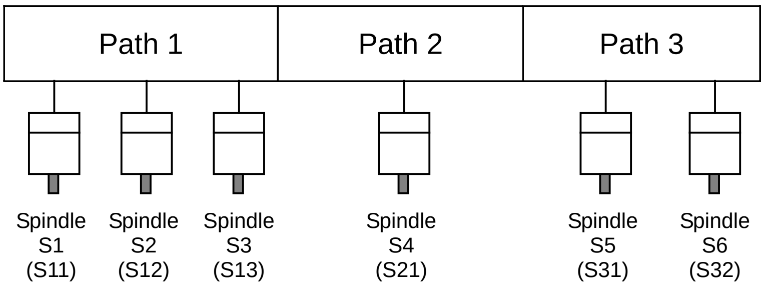


Fig. 16.1.1 (j)

To perform external pulse synchronization with the second spindle (S32) in path 3, set the spindle number of the serial spindle to 6.

NOTE

1. The spindle number to be set does not depend on the path and is common to the system.
2. If the set serial spindle is not present, external pulse synchronization is disabled.
3. Be sure to set EID8g to EID31g (G147 to G149 for group 1) to 0.

Related parameters

Bit 3 (DIAx) of parameter No. 1006
Parameter No. 1424
Bit 1 (CDI) of parameter No. 8005
Bit 3 (ESY) of parameter No. 8007
Bit 0 (EOS) of parameter No. 8019

⚠ **CAUTION**

The feedrate is clamped to the manual rapid traverse rate for each axis (parameter No. 1424).

NOTE

For diameter specification based on bit 3 (DIAx) of parameter No. 1006, whether to specify a radius value or diameter value can be chosen using bit 1 (CDI) of parameter No. 8005. At this time, set the same value with bit 3 (DIAx) of parameter No. 1006 for all axes that belong to the same group.

(13) External pulse synchronization - first manual handle (0Dh)
(14) External pulse synchronization - second manual handle (0Eh)
(15) External pulse synchronization - third manual handle (0Fh)

Each of these commands performs a synchronous operation with a manual handle.

When a negative external pulse value is specified, a movement is made in the opposite direction.
The remaining amount of travel is 0 at all times.
If a manual handle interrupt operation is performed for the same axis, an amount of travel based on external pulses added to manual handle interrupt pulses results. A parameter-set magnification is invalid.

By setting the reset signal ECLRg to “1”, each of these commands can be ended. At this time, the servo motor is decelerated to a stop, the axis moving signal EGENg is set to “0”, and the control axis selection status signal *EAXSL is also set to “0”.
Before specifying the next command, check that the control axis selection status signal *EAXSL is set to “0”. Until the control axis selection status signal *EAXSL is set to “0”, ensure that the reset signal ECLRg continues to be set to “1”.

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	External pulse synchronization - manual handle command (0Dh for the first manual handle) (0Eh for the second manual handle) (0Fh for the third manual handle)
EIF0g to EIF15g	G144,145	Pulse weight

Pulse weight

The specifications of the external pulse synchronization - position coder command (EC0g to EC6g: 0Bh) are applicable.

Related parameters

Same as for external pulse synchronization - position coder

> ⚠ **CAUTION**
> The feedrate is clamped to the manual rapid traverse rate for each axis (parameter No. 1424).

> **NOTE**
> For diameter specification based on bit 3 (DIAx) of parameter No. 1006, whether to specify a radius value or diameter value can be chosen using bit 1 (CDI) of parameter No. 8005. At this time, set the same value with bit 3 (DIAx) of parameter No. 1006 for all axes that belong to the same group.

(16) Speed command (10h)

This command performs a continuous feed operation based on a speed command.

By using bit 0 (ROTx) of parameter No. 1006, set a rotation axis as a controlled axis for which this command is executed.
The continuous feed command (EC0g to EC6g: 06h) exercises position control. However, the speed command (EC0g to EC6g: 10h) can exercise speed control on the servo motor to dynamically change the speed during continuous feed. So, this command is suited, for example, to an application where a servo motor drives a rotary tool.
With parameter No. 8028, 8029 and No. 8032, a time constant for linear acceleration/ deceleration or bell-shaped acceleration/deceleration can be set for each axis.
Every coordinate values are not updated during the speed command.
A reference position return operation needs to be performed after a move command is specified upon completion of the speed command. (This operation also needs when an absolute position detector is attached.)

When bit 2 (VCP) of parameter No. 8007 is set to 0, coordinate value is updated during feed operation based on a speed command. (The FS15 type is used for the speed command in PMC axis control.)
When bit 2 (VCP) of parameter No. 8007 is set to 1, the FS16 type is used for the speed command in PMC axis control. With this setting, the speed command is determined by the setting of bit 4 (EVP) of parameter No. 8005.
When the FS16i type (bit 2 (VCP) of parameter No. 8007 is set to 1) and speed control (bit 4 (EVP) of parameter No. 8005 is set to 0) are specified, coordinates are not updated. Reference position return must be performed before a move command is executed after the completion of the speed command since the machine position is lost. (The machine position is also lost when an absolute position detector is attached.)
When the FS16i type (bit 2 (VCP) of parameter No. 8007 is set to 1) and position control (bit 4 (EVP) of parameter No. 8005 is set to 1) are specified, coordinates are updated.

By setting the reset signal ECLRg to “1”, this command can be ended.
At this time, the servo motor is decelerated to a stop, and the axis moving signal EGENg is set to “0”. Before specifying the next command, check that the axis moving signal EGENg is set to “0”.
Until the axis moving signal EGENg is set to “0”, ensure that the reset signal ECLRg continues to be set to “1”.
After the execution of speed command, to execute a move command in an NC block, execute the method (2) of CAUTION 9 in the item "Caution" described in the end of chapter "PMC CONTROL FUNCTION".

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Speed command (10h)
EIF0g to EIF15g	G144,145	Continuous feedrate

Continuous feedrate

Set the speed of the servo motor with a binary code.

Set a positive value when the servo motor rotates in the forward direction. Set a negative value (two's complement) when the servo motor rotates in the reverse direction.

If a new servo motor speed is set as a feedrate, and the logical state of the axis control command read signal EBUFg is reversed, the servo motor is accelerated or decelerated to the new speed.

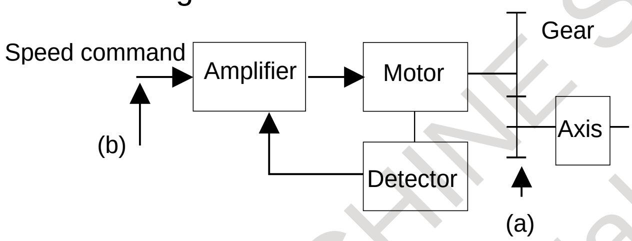
Valid data range	Unit
-32768 to +32767	min^{-1}

Related parameters

Parameter No. 8028
Parameter No. 8029
Parameter No. 8032

NOTE

1 The following feedrate error can occur:



(a) With the speed command based on PMC axis control, a servo motor speed is specified as a feedrate. So, in order to command a feedrate for an axis when a gear is used between the axis and servo motor, the desired feedrate for the axis needs to be converted to a servo motor speed. A feedrate must be specified using an integer, so that when it is converted to a servo motor speed, a rounding error can occur.

(b) The minimum unit of feedrate is given by the expression below. An integer must be specified. No finer value may be specified.
Fmin = P ÷ 1000 (mm/min)
A calculation is made according to IS-B.
Fmin: Minimum feedrate unit
P : Number of pulses per revolution of a detector for speed feedback
A speed is command according to the expression below.
F = N × P ÷ 1000 (mm/min)
A calculation is made according to IS-B.
F : Speed command (integer)
N : Servo motor speed (min^{-1})
P : Number of pulses per revolution of a detector for speed feedback

2 In the speed command mode, a speed after acceleration/deceleration is command for the servo control section. At this time, a position control loop gain is invalid.

3 Machine lock is disabled for the speed command.

Speed command and position control

The PMC axis can be moved at a constant speed by performing the speed command of PMC axis control through position loop control, that is, outputting the command pulses corresponding to a rotation speed to servo position control instead of outputting rotation speed data to the amplifier.

Set the FS16i specification (set bit 2 (VCP) of parameter No. 8007 to 1) and specify either speed loop control or position loop control is used for speed command of PMC axis control using bit 4 (EVP) of parameter No. 8005.

When speed command with position control(bit 4 (EVP) of parameter No. 8005 and bit 2 (VCP) of parameter No. 8007 are set to 1) is selected and uses acceleration/ deceleration, the linear acceleration/deceleration expanded parameter for PMC axis control speed command continuous feed (bit 0 (PTC) of parameter 12730) must be set to 1, and set the linear acceleration/deceleration using following parameters.

Parameter(No.8028):Time constant 1 of linear acceleration/deceleration in feedrate control command under PMC axis control: Unit [msec/1000[min^{-1}]]

Parameter(No.12731):Time constant 2 of linear acceleration/deceleration in feedrate control command under PMC axis control :Unit [msec/1000[min^{-1}]]

Parameter(No.12732):Time constant 3 of linear acceleration/deceleration in feedrate control command under PMC axis control :Unit [msec/1000[min^{-1}]]

Parameter(No.12733):Time constant 4 of linear acceleration/deceleration in feedrate control command under PMC axis control :Unit [msec/1000[min-^{1}]]

Parameter(No.12734):Time constant 5 of linear acceleration/deceleration feedrate control command under PMC axis control :Unit [msec/1000[min^{-1}]]

Parameter(No.12735):1st feedrate for changing time constant of continuous feed operation based on a feedrate control command in PMC axis control :Unit [min^{-1}]

Parameter(No.12736):2nd feedrate for changing time constant of continuous feed operation based on a feedrate control command in PMC axis control :Unit [min^{-1}]

Parameter(No.12737):3rd feedrate for changing time constant of continuous feed operation based on a feedrate control command in PMC axis control :Unit [min^{-1}]

Parameter(No.12738):4th feedrate for changing time constant of continuous feed operation based on a feedrate control command in PMC axis control :Unit [min^{-1}]

When the bell-shaped acceleration/deceleration is used, set the time constant to parameter (No.8029) in addition to the setting of the linear acceleration/deceleration.

For position control (bit 4 (EVP) of parameter No. 8005 is 1), override is enabled.

Set parameter No. 8040 for each axis to the amount of travel per revolution of the motor in terms of the minimum travel unit and convert speed commands to position commands using this value.

As shown in the following block diagram (Fig. 16.1.1 (k)), rotation speed data is not output to the amplifier and the command pulses corresponding to a rotation speed are output to servo position control.

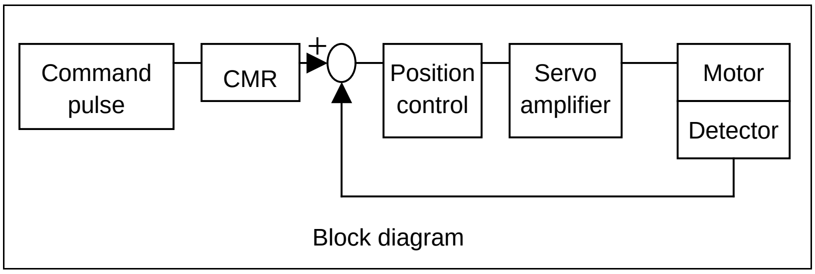


Fig. 16.1.1 (k)

Related parameters

Bit 4 (EVP) of parameter No. 8005
Bit 2 (VCP) of parameter No. 8007
Parameter No. 8040
Bit 0 (PTC) of parameter No. 12730
Parameter No. 12731
Parameter No. 12732

Parameter No. 12733
Parameter No. 12734
Parameter No. 12735
Parameter No. 12736
Parameter No. 12737
Parameter No. 12738

NOTE

1. The commanded control axis to be controlled needs to be set as a rotation axis in bit 0 (ROTx) of parameter No. 1006.
2. When the FS16i specification (bit 2 (VCP) of parameter No. 8007 is 1) and speed control (bit 4 (EVP) of parameter No. 8005 is 0) are specified, coordinates are not updated.
   After execution, the following signal is set to 0 and the position is lost.
   - Reference position establishment signal ZRFx<F120>

   After execution of speed command, establish the reference position.
3. When the FS16i specification (bit 2 (VCP) of parameter No. 8007 is 1) and position control (bit 4 (EVP) of parameter No. 8005 is 1) are specified, coordinates are updated.
4. Only when the FS16i specification (bit 2 (VCP) of parameter No. 8007 is 1) and position control (bit 4 (EVP) of parameter No. 8005 is 1) are specified, feedrate override is enabled.
5. After the execution of speed command, to execute a move command in an NC block, execute the method (2) of CAUTION 9 in the item "Caution" described in the end of chapter "PMC CONTROL FUNCTION".
6. Flexible synchronization control cannot be used together with speed command in PMC axis control (Speed control).

(17) Torque control (11h)

This command performs a continuous feed operation based on torque control.

With this command, position control on a PMC control axis can be switched to torque-based control. The servo motor outputs torque exactly as specified by the NC.

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Torque control command (11h)
EIF0g to EIF15g	G144,145	Maximum feedrate
EID0g to EID31g	G146 to 149	Torque data

Maximum feedrate

Command a maximum feedrate during torque control by using the unit min^{-1}.
When there is no torque generation target or the feedrate exceeds the commanded value during torque control, the alarm SV0422, “EXCESS VELOCITY IN TORQUE” is issued.
When updating the maximum feedrate during torque control set in the torque control mode, set new data on the signals and reverse the logical state of the axis control command read signal EBUFg.

Valid data range	Unit
1 to +32767	min^{-1}

NOTE

When a linear motor is used, the unit of data is cm/min.

Torque data

Command torque data.

Command a positive value when the torque direction is positive. Command a negative value when the torque direction is negative.

When updating the torque data set in the torque control mode, set new data on the signals and reverse the logical state of the axis control command read signal EBUFg.

Valid data range	Unit
-999999999 to +999999999 (9-digit)	0.00001 Nm

NOTE

When a linear motor is used, the unit of data is 0.001N.

(1) Switching from position control to torque control

(a) Torque control axis setting
Using bit 7 (TRQ) of parameter No. 2007, set whether an axis is to be placed under torque control. Moreover, for an axis to be placed under torque control, set the torque constant parameter No. 2105. The standard value of the motor is automatically set in this parameter when the power is turned on after setting bit 1 (DGP) of parameter No. 2000 to 0.

(b) Position management in the torque control mode
Whether to perform follow-up operation in the torque control mode can be chosen using bit 4 (TQF) of parameter No. 1803.
If follow-up operation is performed (with bit 4 (TQF) of parameter No. 1803 set to 1), position control is exercised, regardless of whether the torque control mode is set.
If follow-up operation is not performed (with bit 4 (TQF) of parameter No. 1803 set to 0), the operation depends on whether to update the error counter (bit 1 (TRE) of parameter No. 1805).
If the error counter is updated (with bit 1 (TRE) of parameter No. 1805 set to 0), the alarm SV0423, “EXCESS ERROR IN TORQUE” is issued when the value of the error counter exceeds the value set in parameter No. 1885. At the time of switching back to position control, follow-up operation is performed.
If the error counter is not updated (with bit 1 (TRE) of parameter No. 1805 set to 1), no errors are accumulated, so that no servo alarm is issued. However, when the maximum allowable speed is exceeded, the alarm SV0422, “EXCESS VELOCITY IN TORQUE” is issued.
When this setting is selected, a reference position return operation is required to switch back to position control.

Except when bit 4 (TQF) of parameter No. 1803 is set to 0, and bit 1 (TRE) of parameter No. 1805 is se to 1, the CNC always exercises position control even in the torque control mode. So, no reference position return operation needs to be performed even at the time of switching from torque control to position control.

(c) Movement direction and speed in the torque control mode
In the torque control mode, the output of torque commanded by the axis control data signals EID0g to EID31g is attempted. However, when there is no torque generation target or the output has not reached the commanded torque data yet, a movement is made in the same direction as in the case of position control according to the +/- sign of torque data. When the move speed exceeds the commanded value, the alarm SV0422 is issued.

(d) Timing of switching to torque control
Switching to the torque control mode from another mode occurs when the position deviation is decreased to within the in-position width.

(e) In the torque control mode, the torque control mode signal TRQM1 to TRQM8 is set to “1“. (For details of this signal, see "Signal detail".)

(2) Switching from torque control to position control
(Torque control mode cancellation)
When any of the following conditions occurs, the torque control mode ends:
(i) When the reset signal ECLRg is set to "1"
(ii) When a servo alarm is issued
(iii) When an OT alarm is issued with an axis placed under torque control
(iv) At the time of emergency stop
(v) When servo-off occurs due to the servo-off signal ESOFg

(a) Timing of torque control cancellation
The cancellation of torque control based on PMC axis control depends on whether to perform follow-up operation as determined by bit 4 (TQF) of parameter No. 1803.
- When follow-up operation is performed (with bit 4 (TQF) of parameter No. 1803 set to 1).
The torque control mode signal TRQM1 to TRQM8 is set to 0 to return to position control immediately when a cancellation condition occurs. The execution then stops when the position deviation is decreased to within the in-position width.
- When follow-up operation is not performed (with bit 4 (TQF) of parameter No. 1803 set to 0).
The torque control mode signal TRQM1 to TRQM8 is set to 0 when a cancellation condition occurs. If the error counter is updated (with bit 1 (TRE) of parameter No. 1805 set to 0), the control mode is switched back to position control when the position deviation is decreased to below the value of the cancellation limit parameter No. 1886 after the start of follow-up operation. The execution then ends when the position deviation is decreased to within the in-position width. If the error counter is not updated (bit 1 (TRE) of parameter No. 1805 is set to 1), a reference position return operation must be once performed to return to position control.

(b) Command after cancellation
After the torque control mode is canceled, the control mode is switched to normal position control. In the torque control mode, position is controlled, so that machine coordinates are not shifted. However, a shift occurs between the workpiece coordinates and machine coordinates, so that the shift amount needs to be canceled, for example, by executing workpiece coordinate system preset.
After the execution of torque control, to execute a move command in an NC block, execute the method (2) of CAUTION 9 in the item "Caution" described in the end of chapter "PMC CONTROL FUNCTION".

The Table 16.1.1 (w) indicates the operations dependent on the settings of bit 4 (TQF) of parameter No. 1803 and bit 1 (TRE) of parameter No. 1805.

Table 16.1.1 (w)

Bit 4 (TQF) of parameter No. 1803	Bit 1 (TRE) of parameter No. 1805	Position control	Operation in TRQ mode	Reference position return(*)
1	**0/1**	O	Position control is exercised.	Not required
0	**0**	O	The alarm SV0423 is issued when the error counter exceeds the value of parameter No. 1885.	Not required
0	**1**	-	No SV alarm is issued. When the maximum allowable speed is exceeded, however, the alarm SV0422 is issued.	Required

(*) Whether reference position return operation is required or not when the control mode has been switched from torque control back to position control

TORQUE CONTROL FUNCTION

Set following parameters about servo function "torque control function" properly. As to the detail of the parameters, refer to "FANUC AC SERVO MOTOR αi series PARAMETER MANUAL(B-65270EN)".

Parameter No.2007#7
Parameter No.2203#4
Parameter No.2003#3
Parameter No.2105

If the parameter No.2007#7 or No.2105 is not set correctly, the alarm (DS1184), “PARAMETER ERROR IN TORQUE” occurs.

⚠ **CAUTION**

1 If a movement is made along a torque control axis in the torque control mode, set bit 4 (TQF) of parameter No. 1803 for follow-up operation to 1.
2 If a movement has been made along a torque control axis when torque control is canceled, a mechanical shock occurs at the time of return to position control. As a greater speed is used, a larger shock occurs. So, decelerate to a maximum allowable extent or stop before canceling torque control.
3 When commanding torque control after completion of manual reference position return, set the feed direction selection signal to “0“ or switch to a mode other than the manual reference position return mode beforehand.
4 When the control axis selection signal EAXx is switched in the torque control mode, the alarm PS0139, “CANNOT CHANGE PMC CONTROL AXIS” is issued. Before switching the control axis selection signal EAXx, cancel the torque control mode.
5 In the torque control mode, do not detach a controlled axis with the control axis detach signal DTCHx or a setting parameter.
6 When servo-off occurs, the torque control mode is canceled. However, the torque control state remains to be set. So, be sure to set the reset signal ECLRg to “1”.
7 Machine lock is disabled for the torque control command.
8 In case bit 4 (TQF) of parameter No. 1803 is 0 (Follow-up operation is not performed.) and bit 1 (TRE) of parameter No. 1805 is 1 (Servo error counter is not updated.), after execution of torque control, establish the reference position.
9 After the execution of torque control, to execute a move command in an NC block, execute the method (2) of CAUTION 9 in the item "Caution" described in the end of chapter "PMC CONTROL FUNCTION".

Related parameters

Bit 4 (TQF) of parameter No. 1803
Bit 1 (TRE) of parameter No. 1805
Parameter No. 1885
Parameter No. 1886
Parameters about servo function "torque control function" (See above description)

Related signals

Control axis detach signal DTCHx <G124>

(18) Auxiliary function 1 (12h)
(19) Auxiliary function 2 (14h)
(20) Auxiliary function 3 (15h)

Each of these commands performs the same operation as an auxiliary function (M code Function) of the CNC.

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Auxiliary function command (12h for the auxiliary function 1 command) (14h for the auxiliary function 2 command) (15h for the auxiliary function 3 command)
EID0g to EID15g	G146 to 147	Auxiliary function code

Auxiliary function code

Specify an auxiliary function code to be sent to the PMC with a binary code.
With bit 6 (AUX) of parameter No. 8001, specify a 1-byte or 2-byte code on the signals EID0g to EID15g.

Related parameters

Bit 6 (AUX) of parameter No. 8001

Related signals

Auxiliary function completion signal EFINg <G142.0>
Auxiliary function strobe signal EMFg <F131.0> (for auxiliary function 1)
Auxiliary function 2 strobe signal EMF2g <F131.2> (for auxiliary function 2)
Auxiliary function 3 strobe signal EMF3g <F131.3> (for auxiliary function 3)
Auxiliary function code signals EM11g to EM48g <F132, F142>
Auxiliary function executing signal EDENg <F130.3>

> **NOTE**
> The same auxiliary function cannot be concurrently executed with some groups (For example, command of 12h to 2 groups). Execute a different auxiliary function in each group or execute with another group after the auxiliary function ends.

(21) Machine coordinate system selection (20h)

This command performs the same operation as “G53” of the CNC.

This command performs rapid traverse in the absolute mode to a commanded position in the machine coordinate system. So, this command can be used to move the tool to a specified machine-specific position such as a tool change position.

> **NOTE**
> 1 When using this command with the T series, cancel tool offset and tool-nose radius compensation beforehand.
> 2 When using this command with the M series, cancel tool radius compensation, tool length compensation, and tool offset beforehand.
> 3 Before this command can be specified, a machine coordinate system must be set. So, a manual reference position return operation or reference position return operation based on G28 needs to be performed at least once after the power is turned on. When an absolute-position detector is attached, machine position data is stored, so that a reference position return operation is unnecessary after the power is turned on.
> 4 This command can be specified regardless of the setting (bit 0 (NWZ) of parameter No. 8136) of the workpiece coordinate system.

When the machine coordinate system selection (20h) is commanded with the PMC axis control for the rotary axis to which the roll-over function is valid (bit 0 (ROAx) of parameter No.1008 is set to 1), the direction of the rotation for an absolute command is as follows.

		Bit 4 (R20x) of parameter No.8013	
		0	1
Bit 1 (RABx) of parameter No.1008	0	Direction of the shortest path	Direction of the shortest path
	1	Direction of sign of the amount of the movement to be made	Direction of sign of the command value

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Machine coordinate system selection command (20h)
EIF0g to EIF15g	G144,145	Rapid traverse rate
EID0g to EID31g	G146 to 149	Machine coordinate value

Rapid traverse rate

The specifications of the rapid traverse command (EC0g to EC6g: 00h) are applicable.

Machine coordinate value

For a linear axis, specify a machine coordinate value in the input unit for the axis as an absolute value with a binary code.

Example: For absolute "10000"

Input unit	inch	1.0000	mm	10.000
Output unit	mm	25.400	inch	0.3937

For a rotation axis, a move direction can be selected with a parameter.

Set bit 0 (ROAx) of parameter No. 1008 to 1 to enable the rollover function. Then, select the sign of a commanded value or the shortcut direction with bit 1 (RABx) of parameter No. 1008. Additionally, the amount of travel per revolution needs to be set in parameter No. 1260.

Related parameters

Bit 0 (ROTx) of parameter No. 1006
Bit 0 (ROAx) of parameter No. 1008
Bit 1 (RABx) of parameter No. 1008
Parameter No. 1260

(22) Cutting feed-sec/block specification (21h)

This command performs cutting feed according to a specified period of time. In the sec/block specification, specify a period of time required for ending a block.

Axis control block data

Signal abbreviation	Signal address (group 1)	Data
EC0g to EC6g	G143.0 to G143.6	Cutting feed - sec/block command specification (21h)
EIF0g to EIF15g	G144,145	Cutting feed time
EID0g to EID31g	G146 to 149	Total moving distance

Cutting feed time

Specify a period of time required for ending a block.

[Unit of data] 0.1sec
[Valid data range] 1 to 32767

Total moving distance
The specifications of the rapid traverse command (EC0g to EC6g: 00h) are applicable.

> **NOTE**
> The dry run signal and the override signal are invalid, but the tool stops only when the override signal specifies 0%.

Related parameters and related signals
Basically same as for cutting feed -feed per minute

Notes on the commands

- Display of the remaining amount of travel

When the continuous feed command (EC0g to EC6g: 06h) and the external pulse synchronization commands (EC0g to EC6g: 0Bh, 0Dh to 0Fh) are specified, the remaining amount of travel indicates 0 at all times.

- Command buffering

The continuous feed command (EC0g to EC6g: 06h) and the external pulse synchronization commands (EC0g to EC6g: 0Bh, 0Dh to 0Fh) are not buffered. So, the axis control command read completion signal EBSYg need not be checked at usual command specification time.
When the continuous feed command (EC0g to EC6g: 06h) is specified, the feedrate is changed to a specified value by specifying a value with EIFg and reversing EBUFg. (However, the direction remains unchanged.)

- Acceleration/deceleration

The valid acceleration/deceleration type varies from one command to another.
See Table 16.1.1 (x), "Valid acceleration/deceleration types (for each command)".

Table 16.1.1 (x) Valid acceleration/deceleration types (for each command)

Command	Operation	Acceleration/deceleration type
00h 05h 07h 08h 09h 0Ah 20h	Rapid traverse Reference position return 1st reference position return 2nd reference position return 3rd reference position return 4th reference position return Machine coordinate system selection	The settings of rapid traverse parameters of exponential type, linear type after interpolation, and bell-shaped type after interpolation used with the NC are valid.
01h 02h 0Bh 0Dh 0Eh 0Fh 21h	Cutting feed - feed per minute Cutting feed - feed per revolution External pulse - Position coder - 1st manual handle - 2nd manual handle - 3rd manual handle Cutting feed - sec / block	The settings of cutting parameters of exponential type, linear type after interpolation, and bell-shaped type after interpolation used with the NC are valid. However, when exponential type and value is set in parameters Nos. 8030 and 8031, these value are valid instead of parameters Nos. 1622 and 1623.
06h	Continuous feed	The settings of jog feed parameters of exponential type, linear type after interpolation, and bell-shaped type after interpolation used with the NC are valid. However, when exponential type and value is set in parameters Nos. 8030 and 8031, these value are valid instead of parameters Nos. 1624 and 1625.
10h	Speed command	Set linear-type parameters Nos.8028 and 8032. Set bell-shaped type parameters Nos.8028, 8029 and 8032.

- Immediate commands

The commands listed below are not buffered. The commands are referred to as immediate commands.

(1) Continuous feed(EC0g to EC6g : 06h)
(2) External pulse synchronization - Position coder (EC0g to EC6g : 0Bh)
(3) External pulse synchronization - 1st manual handle (EC0g to EC6g : 0Dh)
(4) External pulse synchronization - 2nd manual handle (EC0g to EC6g : 0Eh)
(5) External pulse synchronization - 3rd manual handle (EC0g to EC6g : 0Fh)
(6) Speed command(EC0g to EC6g : 10h)
(7) Torque control (EC0g to EC6g : 11h)

When an immediate command is prepared and the axis control command read signal EBUFg is reversed during execution of the same type of command, the command being executed is terminated, and the next command is executed immediately.
Suppose, for example, that when continuous feed is being executed at 100 mm/min, another continuous feed command with only the feedrate changed to 300 mm/min is specified and the axis control command read signal EBUFg is reversed. The feedrate changes to 300 mm/min.

To end an immediate command, input the reset signal ECLRg.
When specifying a command other than the immediate commands, set the reset signal ECLRg to “1” and check that the command is terminated, then reverse EBUFg.
(If both of the block being executed and a block being prepared are immediate commands, but the types of commands differ from each other, set the reset signal ECLRg to “1” once.)

- To detect an alarm for unselected PMC axis control

If the controlled axis selection signals EAX1 to EAX8 are “0“, execution of PMC axis control command is ignored. When the bit 2 (EZC) of parameter No.8019 is set to 1, controlled axis selection signals EAX1 to EAX8 state are “0”, alarm DS1451, “IMPROPER PMC AXIS COMMAND” is detected when the PMC axis control command is executed. Unexpected operation of the machine can be prevented by alarm stop.
Alarm DS1451 is detected in the path that belongs to group of PMC axis control. Automatic operation of the path is stopped, operation of PMC axis control of the group is stopped. Further, in the alarm signal AL<Fn001.0> and alarm signal EIALg <F130.2, F133.2, F136.2, F139.2> is set to “1”. If PMC axis control command from the different path, stop the automatic operation by setting the bit 1 (IAL) of parameter No.8100 to 0.

- To detect an alarm for incorrect command after PMC axis control

After the move command of PMC axis control is interrupted by reset signal ECLRg, execution of next command requires the following operations.
(1) Reset signal ECLRg is set to “1”.
(2) Until the control axis selection status signal *EAXSL is set to “0“, ensure that the reset signal ECLRg continues to be set to “1”.
(3) Execution of next PMC axis control command.

Without waiting for the condition (2), the following command can not be executed. When the bit 1 (PIA) of parameter No.8019 is set to 1, As shown in Table 16.1.1 (y) and Table 16.1.1 (z), alarm DS1451, ”IMPROPER PMC AXIS COMMAND” is generated by different acceleration/deceleration type command during deceleration. Incorrect behavior can be suppressed by alarm generating DS1451.

Table 16.1.1 (y) PMC axis control command and acceleration/deceleration type

PMC axis control command	Acceleration/deceleration type
Rapid traverse(00h) Reference position return(05h) First reference position return(07h) Second reference position return(08h) Third reference position return(09h) Fourth reference position return(0Ah) Machine coordinate system selection(20h)	Acceleration/deceleration in rapid traverse
Cutting feed - feed per minute(01h) Cutting feed - feed per revolution(02h) External pulse synchronization - position coder(0Bh) External pulse synchronization - first manual handle(0Dh) External pulse synchronization - second manual handle(0Eh) External pulse synchronization - third manual handle(0Fh) Cutting feed - sec/block specification(21h)	Acceleration/deceleration in cutting feed
Continuous feed(06h)	Acceleration/deceleration in jog feed
Skip - feed per minute(03h)	Acceleration/deceleration disabled

Table 16.1.1 (z) Alarm generated by combination of acceleration /deceleration type (parameter PIA (No.8019#1)=1)

Reset PMC axis control command acceleration/deceleration type	Next PMC axis control command acceleration/deceleration type			
	Rapid traverse	Cutting feed	Jog feed	Disabled
Acceleration/deceleration in Rapid traverse	Commendable	Alarm	Alarm	Alarm
Acceleration/deceleration in Cutting feed	Alarm	Commendable	Alarm	Alarm
Acceleration/deceleration in Jog feed	Alarm	Alarm	Commendable	Alarm

Signal

Signal list (PMC axis control)

Number	Symbol	Signal name
(1)	EAX1 to EAX8	Controlled axis selection signals
(2)	EC0g to EC6g	Axis control command signals
(3)	EIF0g to EIF15g	Axis control feedrate signals
(4)	EID0g to EID31g	Axis control data signals
(5)	EBUFg	Axis control command read signal
(6)	EBSYg	Axis control command read completion signal
(7)	ECLRg	Reset signal
(8)	ESTPg	Axis control temporary stop signal
(9)	ESBKg	Block stop signal
(10)	EMSBKg	Block stop disable signal
(11)	EM11g to EM48g	Auxiliary function code signals
(12)	EMFg	Auxiliary function strobe signal
(13)	EMF2g	Auxiliary function 2 strobe signal
(14)	EMF3g	Auxiliary function 3 strobe signal
(15)	EFINg	Auxiliary function completion signal
(16)	ESOFg	Servo-off signal
(17)	EMBUFg	Buffering disable signal
(18)	*EAXSL	Control axis selection status signal
(19)	EINPg	In-position signal
(20)	ECKZg	Following zero checking signal
(21)	EIALg	Alarm signal
(22)	EGENg	Axis moving signal
(23)	EDENg	Auxiliary function executing signal
(24)	EOTNg	Negative-direction overtravel signal

Number	Symbol	Signal name
(25)	EOTPg	Positive-direction overtravel signal
(26)	*EFOV0g to *EFOV7g	Feedrate override signals
(27)	*EROV0g to *EROV7g	1% step rapid traverse override signals
(28)	EOVCg	Override cancellation signal
(29)	EROV1, EROV2	Rapid traverse override signals
(30)	EDRN	Dry run signal
(31)	ERT	Manual rapid traverse selection signal
(32)	EOV0	Override 0% signal
(33)	ESKIP	Skip signal
(34)	EADEN1 to EADEN32	Distribution completion signals
(35)	EABUFg	Buffer full signal
(36)	EACNT1 to EACNT32	Controlling signals
(37)	ELCKZg	Accumulated zero check signal
(38)	TRQM1 to TRQM8	Torque control mode signal

Signal list (Related signals)

Number	Symbol	Signal name
(1)	*+ED1 to *+ED8 *- ED1 to *- ED8	External deceleration signal 1
(2)	*+ED21 to *+ED28 *- ED21 to *- ED28	External deceleration signal 2
(3)	*+ED31 to *+ED38 *- ED31 to *- ED38	External deceleration signal 3
(4)	*+ED41 to *+ED48 *- ED41 to *- ED48	External deceleration signal 4
(5)	*+ED51 to *+ED58 *- ED51 to *- ED58	External deceleration signal 5
(6)	HROV	1% rapid traverse override select signal
(7)	*HROV0 to *HROV6	1% rapid traverse override signals
(8)	FHROV	0.1% rapid traverse override select signal
(9)	*FHRO0 to *FHRO9	0.1% rapid traverse override signals
(10)	ROV1, 2	Rapid traverse override signals
(11)	ZP1 to ZP8	First reference position return completion signal
(12)	ZP21 to ZP28	Second reference position return completion signal
(13)	ZP31 to ZP38	Third reference position return completion signal
(14)	ZP41 to ZP48	Fourth reference position return completion signal
(15)	NDCAL1 to NDCAL8	A/B phase detector disconnection alarm ignore signal
(16)	HNDMP	Manual pulse magnification change signal

Signal address

The method of counting the addresses of the input/output signals G and F is described below. (See Table 16.1.1 (b), "Allocation of group-by-group signals".)

For group-by-group signals, the addresses of group 5 and up are determined in such a way that 1000 is incremented for every four groups.

For axis-by-axis signals, the addresses of the 9th axis and up are determined in such a way that 1000 is incremented for every eight axes.

For axis-by-axis signals in a path, the addresses for path 2 and up are determined in such a way that 1000 is incremented for each path.

Example:

The address of the EBUFg signal for group 10 is G2154.7.

The address of the EAX signal is determined as follows:

Path 1:	1st axis	G136.0
	2nd axis	G136.1
	3rd axis	G136.2
Path 2:	1st axis	G1136.0
	2nd axis	G1136.1

Signal Detail

Each signal is detailed below.
"<>" in the title of each signal indicates a signal address. For group-by-group signals, addresses for group 1 to group 4 are indicated. For axis-by-axis signals, addresses for the 1st axis to 8th axis are indicated.

Signal Detail (PMC axis control)

(1) Controlled axis selection signals EAX1 to EAX8 <Gn136>

[Classification] Input signal, axis-by-axis signal

[Function] Exercises PMC axis control.

[Function] When the signal is set to “1”, PMC axis control becomes valid.
When the signal is set to “0”, PMC axis control becomes invalid. Changing the setting of the control axis selection signal is possible only when control axis selection status signal *EAXSL is set to “0”. Changing the setting when *EAXSL is set to “1” results in the issue of an alarm PS0139, “CANNOT CHANGE PMC CONTROL AXIS”. Alarm signal EIALg is set to “1”. When the bit 5 (NCC) of parameter No. 8001, is set to 0, a command issued from the CNC is executed while the control axis selection signal is set to “1” and signal *EAXSL is set to “0”. When the parameter is set to “1”, the same attempt results in the issue of an alarm PS0139.
If the control axis selection signal is set to “1” while the CNC is currently executing a command, an alarm is generated.
While *EAXSL is set to “0”, the status of alarm signal EIALg does not change to “1” when the control axis selection signal is set to “1” and an alarm PS0139 is generated. In this case, the axis can be controlled from the PMC, even when the CNC is in the alarm status.

> **NOTE**
> After setting control axis selection signals EAX1 to EAX8 to “1”, it takes at least 8 msec before the PMC can issue commands to the CNC.

(2) Axis control command signals EC0g to EC6g<G143.0 to 143.6, G155.0 to 6, G167.0 to 6, G179.0 to 6>

[Classification] Input signal, group-by-group signal

[Function] One of the axis control block data signals

[Operation] Specifies a type of command with 7 bits.
For the meaning of each command, see Table 16.1.1 (g), "Command list".

(3) Axis control feedrate signals EIF0g to EIF15g <G144 to 145, G156 to 157, G168 to 169, G180 to 181>

[Classification] Input signal, Group-by-group signal

[Function] One of the axis control block data signals

[Operation] Two-byte command interface area
Each command has a different function. For details, see Table 16.1.1 (g), "Command list".

(4) Axis control data signals
EID0g to EID31g <G146 to 149, G158 to 161, G170 to 173, G182 to 185>

[Classification] Input signal, group-by-group signal

[Function] One of the axis control block data signals

[Operation] Four-byte command interface area
Each command has a different function. For details, see Table 16.1.1 (g), "Command list".

(5) Axis control command read signal EBUFg <G142.7, G154.7, G166.7, G178.7>

[Classification] Input signal, group-by-group signal

[Function] Directs the CNC to read a block of command data for PMC axis control.

[Operation] For the operation and procedure applicable when the level of this signal is changed from “0” to “1” or from “1” to “0”, see Fig. 16.1.1 (l), "Timing chart of command operation" and Table 16.1.1 (f), "Buffering status in PMC axis control".

(6) Axis control command read completion signal
EBSYg <F130.7, F133.7, F136.7, F139.7>

[Classification] Output signal, group-by-group signal

[Function] Notifies the system that the CNC has read a block of command data for PMC axis control and has stored the block in the input buffer.

[Operation] For the operation and procedure applicable when the level of this signal is changed from “0” to “1” or from “1” to “0”, see Fig. 16.1.1 (m), "Timing chart of command operation" and Table 16.1.1 (f), "Buffering status in PMC axis control".

(7) Reset signal ECLRg <G142.6, G154.6, G166.6, G178.6>

[Classification] Input signal, group-by-group signal

[Function] Resets a PMC axis control command.

[Operation] Resets the corresponding PMC-controlled axis.
When this signal is set to “1”, the following is performed:

(1) When the tool is moving along the axis: decelerates and stops the tool.
(2) When the tool is dwelling: Stops the operation.
(3) When an auxiliary function is being executed: Stops the operation.
Simultaneously, all buffered commands are canceled. Any control command is ignored while this signal is set to “1”.

The continuous feed command (EC0g to EC6g: 06h) and external pulse synchronization command (EC0g to EC6g: 0Bh, 0Dh to 0Fh) can be terminated by setting reset signal ECLRg to “1”. When these commands are terminated, the servo motor decelerates and stops, the axis moving signal EGENg is set to “0”, and the controlled axis selection status signal *EAXSL is set to “0”.
Confirm that the controlled axis selection status signal *EAXSL has been set to “0” before issuing the next command. Do not set reset signal ECLRg to “0” until the controlled axis selection status signal *EAXSL has been set to “0”.

The speed command (EC0g to EC6g: 10h) can also be terminated by setting the reset signal ECLRg to “1”. When this command is terminated, the servo motor decelerates and stops, and the axis moving signal EGENg is set to “0”. Confirm that the axis moving signal EGENg has been set to “0” before issuing the next command. Do not attempt to set the reset signal ECLRg to “0” until the axis moving signal EGENg has been set to “0”.

(8) Axis control temporary stop signal ESTPg <G142.5, G154.5, G166.5, G178.5>

[Classification] Input signal, group-by-group signal

[Function] Temporarily stops a movement before the execution of a block is completed.

[Operation] When this signal is set to “1”, the following is performed:

(1) When the tool is moving along the axis: Decelerates and stops the tool.
(2) When the tool is in dwell: Stops the operation.
(3) When an auxiliary function is being executed: Stops the operation when auxiliary function completion signal EFINg is input.

The stopped operation can be restarted by setting this signal to “0”.

(9) Block stop signal ESBKg <G142.3, G154.3, G166.3, G178.3>
(10)Block stop disable signal EMSBKg <G143.7,G155.7, G167.7, G179.7>

[Classification] Input signal, group-by-group signal

[Function] Enables a stop for each command block, or disables a stop for each block.

[Operation] When block stop signal ESBKg is set to “1” during the execution of a command issued from the PMC, axis control is stopped after the block being executed is completed. When this signal is set to “0”, the buffered command is executed. Block stop signal ESBKg is disabled when block stop disable signal EMSBKg is set to 1 for the block.

Fig. 16.1.1 (l), "Timing chart of block stop related signals" shows the timing chart of command operation.

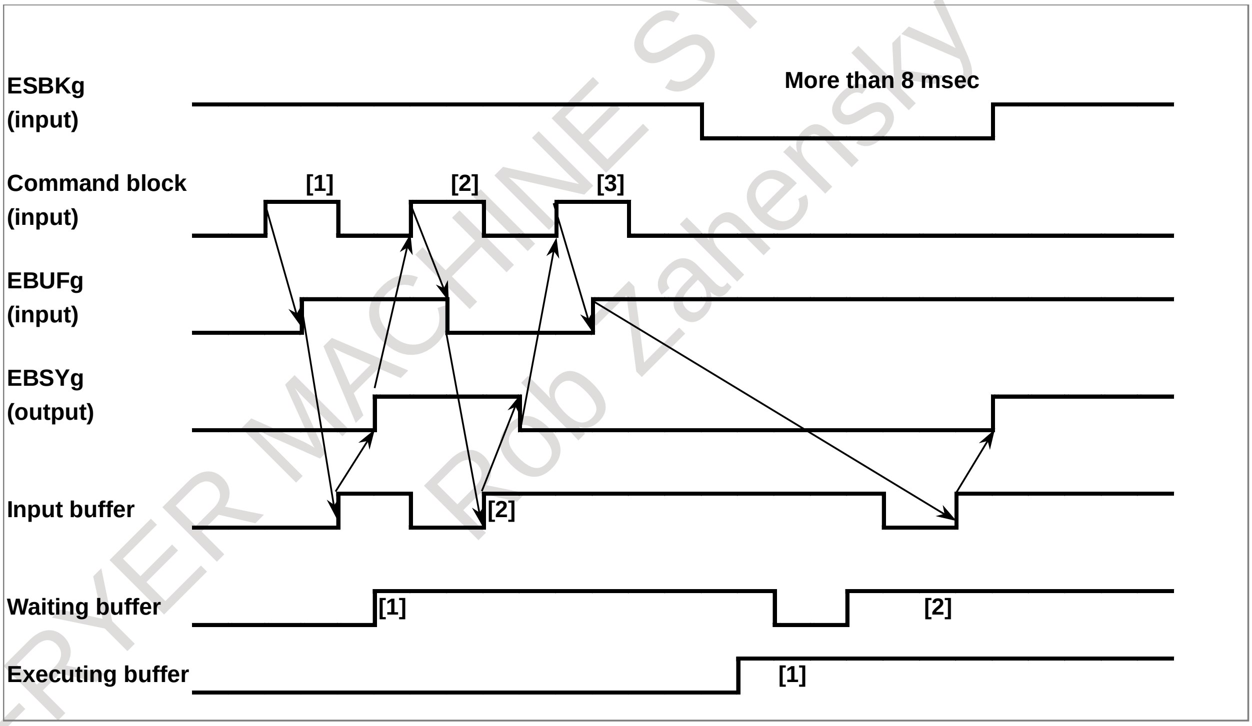


Fig. 16.1.1 (l) Timing chart of block stop related signals

(11) Auxiliary function code signals
EM11g to EM48g <F132, F142, F135, F145, F138, F148, F141, F151>

[Classification] Output signal, group-by-group signal

(12) Auxiliary function strobe signal EMFg <F131.0, F134.0, F137.0, F140.0>

[Classification] Output signal, group-by-group signal

(13) Auxiliary function 2 strobe signal EMF2g <F131.2, F134.2, F137.2, F140.2>

[Classification] Output signal, group-by-group signal

(14) Auxiliary function 3 strobe signal EMF3g <F131.3, F134.3, F137.3, F140.3>

[Classification] Output signal, group-by-group signal

(15) Auxiliary function completion signal EFINg <G142.0, G154.0, G166.0, G178.0>

[Classification] Input signal, group-by-group signal

[Function] Completes an auxiliary function

[Operation] When an auxiliary function command (EC0g to EC6g: 12h) is issued by the PMC, the auxiliary function code is specified in a byte (using signals EID0g to EID7g) or two bytes (using signals EID0g to EID15g), depending on the setting of bit 6 (AUX) of parameter No. 8001.

The CNC sends the auxiliary function code specified in signals EID0g to EID7g and EID8g to EID15g to auxiliary function code signals EM11g to EM28g and EM31g to EM48g and awaits the auxiliary function completion signal EFINg. When the auxiliary function completion signal EFINg is returned, the CNC starts executing the next block.

The timings for sending the auxiliary function code signals and auxiliary function strobe signal, as well as for receiving the auxiliary function completion signal, are the same as those for the auxiliary functions (M functions) under the control of the CNC. See "Auxiliary function executing signal" for details.

(16) Servo-off signal ESOFg <G142.4, G154.4, G166.4, G178.4>

[Classification] Input signal, group-by-group signal

[Function] Changes the servo-off state.

[Operation] When this signal is set to “1”, the servo motor for the corresponding PMC-controlled axis is turned off (servo-off state). When this signal is set to “0”, the servo motor is turned on.

When a torque control command (EC0g to EC6g: 11h) is specified, entering the servo-off state cancels torque control mode, but the torque control state remains set. In such a case, set reset signal ECLRg to “1”.

(17) Buffering disable signal EMBUFg <G142.2, G154.2, G166.2, G178.2>

[Classification] Input signal, group-by-group signal

[Function] Changes the buffering disabled state.

[Operation] When this signal is set to “1”, commands from the PMC are not read while the executing, waiting, or input buffer contains a block. If this signal is set to “1” when any of these buffers contain a block, that block is executed but subsequent commands are read only when the buffers are all empty.

To discriminate the buffering disabled condition, the CNC outputs the axis control command read completion signal (EBSYg) only when a command is read when all buffers are empty.

Fig. 16.1.1 (m), "Timing chart of buffering related signals" shows the timing chart of command operation.

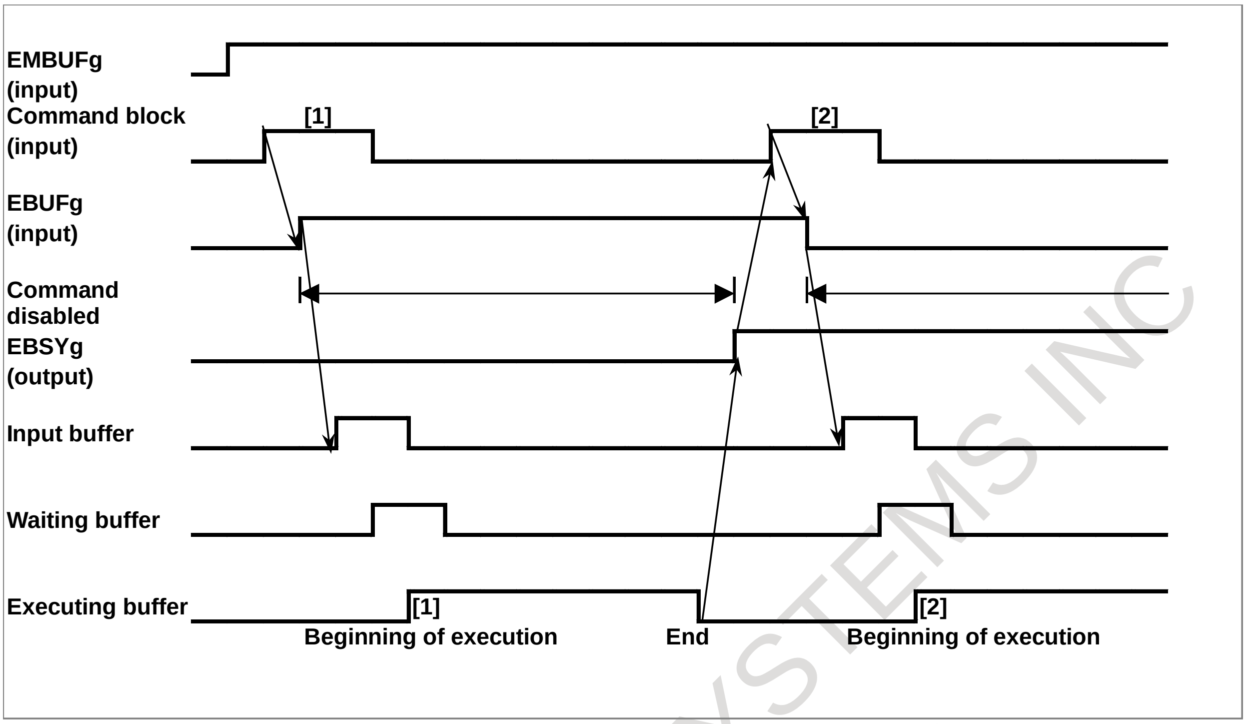


Fig. 16.1.1 (m) Timing chart of buffering related signals

Buffering is disabled, regardless of the buffering disable signal EMBUFg, for the following commands:
(1) Skip-feed per minute (EC0g to EC6g : 03h)
(2) Reference position return (EC0g to EC6g : 05h)
(3) 1st reference position return (EC0g to EC6g : 07h)
(4) 2nd reference position return (EC0g to EC6g : 08h)
(5) 3rd reference position return (EC0g to EC6g : 09h)
(6) 4th reference position return (EC0g to EC6g : 0Ah)
(7) Machine coordinate system selection (EC0g to EC6g : 20h)

---

(18) Controlled axis selection status signal *EAXSL <F129.7>

[Classification] Output signal, path-by-path signal

[Function] Indicates whether PMC axis control is being exercised.

[Operation] When this signal is set to “0”, controlled axis selection signals EAX1 to EAX8 can be changed.

This signal is set to “1” in the following cases:
(1) When the tool is moving along a PMC-controlled axis
(2) When a block is being read into a buffer
(3) When the servo-off signal ESOFg is set to “1”

When this signal is set to “1”, controlled axis selection signals EAX1 to EAX8 cannot be changed. Any attempt to change these signals results in the output of an alarm PS0139, “CANNOT CHANGE PMC CONTROL AXIS”.

If an attempt to change signals EAX1 to EAX8 is made when servo-off signal ESOFg is “1”, an alarm PS0139 occurs and cannot be released simply by setting reset signal ECLRg to “1”. In such a case, restore signals EAX1 to EAX8 or set servo-off signal ESOFg to “0” before setting reset signal ECLRg to “1”. When a command is issued for any of the four groups with PMC axis control, signal *EAXSL is set to “1” to disable axis selection. Thus, changing signals EAX1 to EAX8 results in the output of an alarm PS0139. For groups for which commands are not issued, however, axis selection is enabled if bit 5 (DSL) of parameter No. 8004 is set accordingly.

(19) In-position signal EINPg <F130.0, F133.0, F136.0, F139.0>

[Classification] Output signal, group-by-group signal

[Function] Indicates the in-position state.

[Operation] This signal is set to “1” when the corresponding PMC-controlled axis is in the in-position state.
When the tool is decelerated, in-position check is performed to disable the next command until the tool enters the in-position area. The in-position check, however, can be skipped using bit 6 (NCI) of parameter No. 8004 to reduce the cycle time.

> **NOTE**
> When the axis is fed at a very low speed, the in -position signal might turn to “1”.

(20) Following zero checking signal ECKZg <F130.1, F133.1, F136.1, F139.1>

[Classification] Output signal, group-by-group signal

[Function] Indicates the following zero state.

[Operation] This signal is set to “1” when following zero check or in-position check is being performed for the corresponding PMC-controlled axis.
The following zero state means that the acceleration/deceleration delay amount is zero.

(21) Alarm signal EIALg <F130.2, F133.2, F136.2, F139.2>

[Classification] Output signal, group-by-group signal

[Function] Indicates the alarm state related to PMC axis control.

[Operation] This signal is set to “1” when a servo alarm, overtravel alarm, or alarm PS0130, “NC AND PMC AXIS ARE CONFLICTED” or PS0139, “CANNOT CHANGE PMC CONTROL AXIS” occurs for the corresponding PMC-controlled axis. This signal is set to “0“ when the reset signal ECLRg is set to “1” after the alarm is released, as described below.

Servo alarm

Eliminate the cause of the alarm, then reset the CNC.

Overtravel alarm

Move the tool into the area within the stored stroke limit, then reset the CNC.
The following commands can be used to move the tool into the area within the stored stroke limit during an overtravel alarm:

(1) Rapid traverse (EC0g to EC6g : 00h)
(2) Cutting feed - feed per minute (EC0g to EC6g : 01h)
(3) Cutting feed - feed per rotation(EC0g to EC6g : 02h)
(4) Continuous feed (EC0g to EC6g : 06h)
(5) External pulse synchronization - first manual handle (EC0g to EC6g : 0Dh)
(6) External pulse synchronization - second manual handle (EC0g to EC6g : 0Eh)
(7) External pulse synchronization - third manual handle (EC0g to EC6g : 0Fh)

Alarm (PS0130, PS0139)

Reset the CNC.
See "Alarms and messages" for details.

Reset signal ECLRg cannot be used to reset the CNC in the above cases. Use the reset button on the setting panel, external reset signal ERS, or emergency stop signal *ESP.

Even if an alarm occurs on an axis not related to the group, alarm signal EIALx does not generally change from “0” to “1”.

(22) Axis moving signal EGENg <F130.4, F133.4, F136.4, F139.4>

[Classification] Output signal, group-by-group signal

[Function] Indicates the state of movement on an axis.

[Operation] This signal is set to “1” when the tool is moving along the corresponding PMC-controlled axis according to commands such as rapid traverse (EC0g to EC6g: 00h) and cutting feed (EC0g to EC6g: 01h).
When the dwell command (EC0g to EC6g: 04h) is specified, this signal remains to be set to “0”.

> **NOTE**
> This signal is set to “0” when distribution for the axis is completed (the signal is set to “0” during deceleration).

(23) Auxiliary function executing signal EDENg <F130.3, F133.3, F136.3, F139.3>

[Classification] Output signal, group-by-group signal

[Function] Indicates the state of auxiliary function execution.

[Operation] When an auxiliary function (EC0g to EC6g: 12h) is specified by the PMC, this signal is set to “1” during the period from when auxiliary function codes EID0g to EID15g are sent to auxiliary function code signals EM11g to EM48g until the auxiliary function completion signal EFINg is returned.
Fig. 16.1.1 (n), "Timing chart of auxiliary function related signals" shows the timing chart of command operation.
TMF and TFIN in the table are set in parameters Nos. 3010 and 3011.

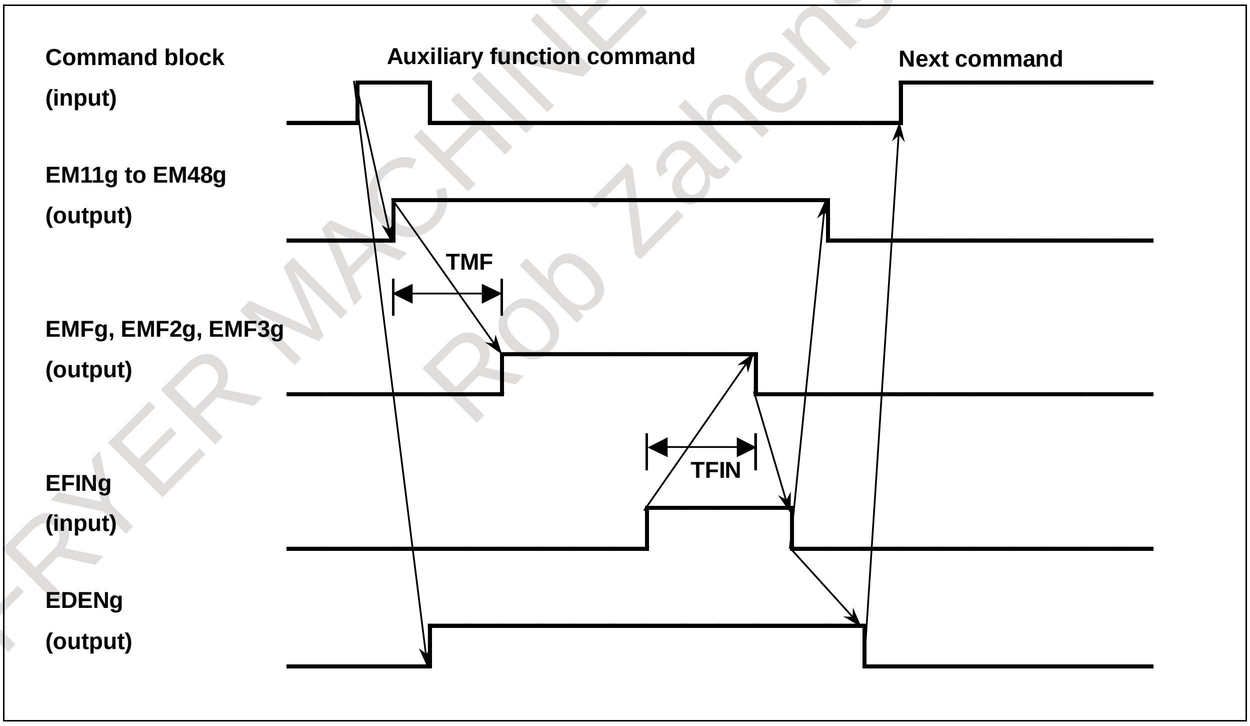


Fig. 16.1.1 (n) Timing chart of auxiliary function related signals

(24) Negative-direction overtravel signal EOTNg <F130.6, F133.6, F136.6, F139.6>
(25) Positive-direction overtravel signal EOTPg <F130.5, F133.5, F136.5, F139.5>

[Classification] Output signal, group-by-group signal

[Function] Indicates the overtravel state.

[Operation] These signals are set to “1” when an overtravel alarm is detected. When the stroke limit in the negative direction is exceeded, signal EOTNg is set to “1”. When the stroke limit in the positive direction is exceeded, signal EOTPg is set to “1”. Simultaneously, alarm signal EIALg is set to “1”.
These signals are set to “0“ when the overtravel alarm is released and reset signal ECLRg is set to “1”. See "Alarm signal EIALg" for details of how to release an overtravel alarm.

(26) Feedrate override signals *EFOV0g to *EFOV7g <G151 (, G163, G175, G187)>

[Classification] Input signal, path-by-path signal (group-by-group signal)

[Function] Applies cutting override.

[Operation] Like the CNC's feedrate override signals *FV0 to *FV7, these signals can be used to select the override for the cutting feedrate, in steps of 1% from 0 to 254%, independently of the CNC using the bit 2 (OVE) of parameter No. 8001.
These signals form an eight-bit binary code and correspond to the override value as follows:

$$\text{Override value} = \sum_{i=0}^{7} \left| 2^{i} \times Vi \right| \%$$

Vi = 0 when signal *EFOVi is “1”
Vi = 1 when signal *EFOVi is “0”
That is, each signal has the following significance:

*EFOV7 = 128%,	*EFOV3 = 8% ,
*EFOV6 = 64%,	*EFOV2 = 4% ,
*EFOV5 = 32%,	*EFOV1 = 2% ,
*EFOV4 = 16%,	*EFOV0 = 1%

When all signals are set to “0”, the override is regarded as being 0%, as well as when all signals are “1”.
Bit 5 (IFV) of parameter No. 8005 is used to determine whether these signals are path-by-path signals or group-by-group signals.
When the IFV parameter is set to 0, these signals are path-by-path signals. When the IFV parameter is set to 1, these signals are group-by-group signals.

(27) 1% step rapid traverse override signals *EROV0g to *EROV7g<G151 (, G163, G175, G187)>

[Classification] Input signal, path-by-path signal (group-by-group signal)

[Function] These signals use the same addresses as the feedrate override signals. When bit 1 (OVR) of parameter No. 8013 is set to 1, these signals are used for rapid traverse override.

[Operation] When bit 2 (OVE) of parameter No. 8001 is set to 1 and bit 1 (OVR) of parameter No. 8013 is set to 1, these signals can be used to select the override for the rapid traverse rate, in steps of 1% from 0% to 100%, independently of the CNC. The override is clamped to 100%.
Each corresponding signal has the following significance:

*EROV7 = 100%	*EROV3 = 8%
*EROV6 = 64%	*EROV2 = 4%
*EROV5 = 32%	*EROV1 = 2%
*EROV4 = 16%	*EROV0 = 1%

For example, to set an override of 5%, set signals *EFOV7 to *EFOV0 to 11111010, which corresponds to a binary code of 00000101.

When bit 7 (IFH) of parameter No. 11850 is set to 0, these signals are used as path-by-path signals. When the bit is set to 1, they are used as group-by-group signals.

> **NOTE**
> When all signals are set to “0”, an override of 100% is assumed, while an override of 0% is assumed for *EFOVg.

(28) Override cancellation signal EOVCg <G150.5 (G162.5, G174.5, G186.5)>

[Classification] Input signal, path-by-path signal (group-by-group signal)

[Function] Disables override.

[Operation] When override is enabled, independently of the CNC, by setting the bit 2 (OVE) of parameter No. 8001, setting this signal to “1” fixes the cutting feed override to 100%. This signal does not affect the rapid traverse override.
Bit 5 (IFV) of parameter No. 8005 is used to determine whether these signals are path-by-path signals or group-by-group signals.
When the IFV parameter is set to 0, these signals are path-by-path signals. When the IFV parameter is set to 1, these signals are group-by-group signals.

(29) Rapid traverse override signals EROV1, EROV2 <Gn150.0,1>

[Classification] Input signal, path-by-path signal

[Function] Applies rapid traverse override.

[Operation] These signals can be used to select the override for the rapid traverse rate, independently of the CNC, by setting of the bit 2 (OVE) of parameter No. 8001.

Rapid traverse override signals		Override value
EROV2	EROV1	
0	0	100%
0	1	50%
1	0	25%
1	1	F0

F0 is the minimum feedrate specified with parameter No. 1421.

(30) Dry run signal EDRN <G150.7>
(31) Manual rapid traverse selection signal ERT <G150.6>

[Classification] Input signal, path-by-path signal

[Function] Applies dry run, or chooses whether to apply dry run.

[Operation] These signals can be used to perform dry run or manual rapid traverse, independently of the CNC, by setting the bit 2 (OVE) of parameter No. 8001. When the dry run signal EDRN is set to “1”, the specified rapid traverse rate and cutting feedrate are ignored and the tool moves at the dry run speed (set in parameter No. 1410) multiplied by the specified manual feedrate override. Bit 3 (RDE) of parameter No. 8001 can be used to specify whether to enable or disable dry run for rapid traverse.
When the manual rapid traverse selection signal ERT is set to “1” during dry run, the tool moves at the rapid traverse rate for rapid traverse and at the maximum jog feedrate for cutting feed. When the signal is set to “0”, the tool moves at the jog feedrate. When the dry run signal EDRN is set to “0”, the specified rapid traverse rate or cutting feedrate is restored.

Manual rapid traverse select signal	Command from PMC	
	At rapid traverse	At Cutting feed
1	Rapid traverse rate	Cutting feedrate
0	Dry run feedrate x JV (NOTE) or Rapid traverse rate *	Dry run feedrate x JV (NOTE)

* Dry run feedrate × JV when bit 3 (RDE) of parameter No. 8001 is 1. Rapid traverse rate when parameter RDE is 0.

> **NOTE**
> When bit 1 (JOV) of parameter No. 1402 is set to 1, the manual feedrate override signal *JV is not applied, but the dry run feedrate is just applied.

---

(32) Override 0% signal EOV0 <F129.5>

[Classification] Output signal, path-by-path signal
[Function] Indicates whether the override value is 0%.
[Operation] This signal is set to “1” when the feedrate override is 0%.

---

(33) Skip signal ESKIP <X0004.6>, ESKIP$^{\#2}$ <X0013.6>, ESKIP$^{\#3}$ <X0011.6>

[Classification] Direct input signal, path-by-path signal
[Function] Applies skip.
[Operation] When this signal is set to “1” during execution of the skip cutting command, the block being executed is immediately stopped and the next block is executed. This signal is unique during PMC axis control. Bit 7 (SKE) of parameter No. 8001 can be used to select whether to use signal SKIP, which is the common skip signal for the PMC and CNC, or PMC-specific skip signal ESKIP.

This skip signal is valid for those axes that are assigned to path 1 to path 3.
For those axes that are assigned to path 1, ESKIP (X004.6) is valid.
For those axes that are assigned to path 2, ESKIP$^{\#2}$ (X013.6) is valid.
For those axes that are assigned to path 3, ESKIP$^{\#3}$ (X011.6) is valid.
This skip signal cannot be used for those axes that are assigned to path 4.

---

(34) Distribution completion signals EADEN1 to EADEN32 <F112>

[Classification] Output signal, axis-by-axis signal
[Function] Indicates the state of distribution based on PMC axis control.
[Operation] These signals are set to “0” when the tool is moving with a command from the PMC. The signals are set to “1” when the tool is not moving, except when it is stopped by an axis control temporary stop signal ESTPg during the execution of a move command.

---

(35) Buffer full signal EABUFg <F131.1, F134.1, F137.1, F140.1>

[Classification] Output signal, group-by-group signal
[Function] Indicates the PMC axis control command buffering state.
[Operation] When the group input buffer holds a command block, this signal is set to “1”. When no command is buffered, this signal is set to “0”.

---

(36) Controlling signals EACNT1 to EACNT32 <F182>

[Classification] Output signal, axis-by-axis signal
[Function] Indicates that PMC axis control is being exercised.
[Operation] When the control axis selection status signal *EAXSL is set to “1”, the bit signal corresponding to an axis being controlled is set to “1”.
A bit signal is set to “1“ also when the servo-off signal ESOFg is set to “1”.

---

(37) Accumulated zero check signal ELCKZg <G142.1, G154.1, G166.1, G178.1>

[Classification] Input signal, group-by-group signal
[Function] Makes an accumulated zero check.
[Operation] Setting this signal to “1” causes an accumulated zero check between blocks to be made at a subsequent cutting feed command. The accumulated zero state means that the acceleration/deceleration delay amount is zero.

(1) Cutting feed - feed per minute (EC0g to EC6g: 01h)
(2) Cutting feed -feed per rotation (EC0g to EC6g: 02h)
(3) Cutting feed - sec/block (EC0g to EC6g : 21h)
This signal can be used, for example, with the chopping function.

---

(38) Torque control mode signal
TRQM1 to TRQM8 <F190>

[Classification] Output signal, axis-by-axis signal

[Function] Indicates an axis in the torque control mode.

[Operation] When this signal is set to "1", which axis is placed in the torque control mode by the torque control command (EC0g to EC6g: 11h) based on PMC axis control is indicated.

Signal detail (Related signals)

The signals related to PMC axis control are detailed below.
The [Function] field and [Operation] field provide descriptions related to PMC axis control. For the general functions of the signals, see the description of each signal.

---

(1) External deceleration signals 1
*+ED1 to *+ED8 <G118>, *-ED1 to *-ED8 <G120>
(2) External deceleration signals 2
*+ED21 to *+ED28 <G101>, *-ED21 to *-ED28 <G103>
(3) External deceleration signals 3
*+ED31 to *+ED38 <G107>, *-ED31 to *-ED38 <G109>
(4) External deceleration signals 4
*+ED41 to *+ED48 <G341>, *-ED41 to *-ED48 <G342>
(5) External deceleration signals 5
*+ED51 to *+ED58 <G343>, *-ED51 to *-ED58 <G344>

[Classification] Input signal, axis-by-axis signal

[Function] Decelerates the movement along a specified axis to a parameter-set feedrate.

[Operation] While a signal is set to "0" for an axis, the feedrate along the axis in the specified direction can be forcibly decelerated to a certain feedrate (external deceleration feedrate) set by a parameter (dependent on the type of command).
If the feedrate is lower than the external deceleration feedrate, however, the specified feedrate continues to be used.
The feedrate for other axes with these signals not set to "0" is not affected.
For details, refer to the specifications of the external deceleration function.
A signal is available for each controlled axis and for each direction. Each signal name includes the sign + or - for direction indication, and the number at the end of each signal name indicates a controlled axis number. These signals are shared by the CNC.

* x ED y z

z
1 : Feed along the first axis is decelerated.
2 : Feed along the second axis is decelerated.
3 : Feed along the third axis is decelerated.

y
None: Follows external deceleration setting 1.
2 : Follows external deceleration setting 2.
3 : Follows external deceleration setting 3.
4 : Follows external deceleration setting 4.
5 : Follows external deceleration setting 5.

x
+ : Feed in the positive direction is decelerated.
- : Feed in the negative direction is decelerated.

When any of the following axis control command is specified while the bit 0 (EDC) of parameter No. 8005 is held at 1, the external deceleration function becomes effective:

(1)	Rapid traverse	(EC0g to EC6g : 00h)
(2)	Cutting feed - feed per minute	(EC0g to EC6g : 01h)
(3)	Reference position return	(EC0g to EC6g : 05h)
(4)	Continuous feed	(EC0g to EC6g : 06h)
(5)	First reference position return	(EC0g to EC6g : 07h)
(6)	Second reference position return	(EC0g to EC6g : 08h)
(7)	Third reference position return	(EC0g to EC6g : 09h)
(8)	Fourth reference position return	(EC0g to EC6g : 0Ah)
(9)	Machine coordinate system selection	(EC0g to EC6g : 20h)
(10)	Cutting feed - sec/block	(EC0g to EC6g : 21h)

For each command above, the external deceleration feedrate settings for axis-by-axis rapid traverse (parameters Nos. 1427, 1441, 1444, 12752, and 12755) are valid.
To cutting feed - feed per minute (EC0g to EC6g: 01h) and cutting feed - sec/block (EC0g to EC6g: 21h) as well, the settings for rapid traverse are applied instead of the external deceleration feedrate settings for cutting feed (parameters Nos. 1426, 1440, 1443, 12751, and 12754).

---

(6) 1% rapid traverse override select signal HROV <G96.7>

[Classification] Input signal, path-by-path signal

---

(7) 1% rapid traverse override signal *HROV0 to *HROV6 <G96.0 to G96.6>

[Classification] Input signal, path-by-path signal
[Function] Enables override to be applied in steps of 1%. Shared by the CNC.
[Operation] If the 1% rapid traverse override select signal HROV <G96.7> is set to “1” when bit 2 (OVE) of parameter No. 8001 is set to 0, rapid traverse override can be specified in steps of 1% by using the 1% rapid traverse override signals *HROV0 to *HROV6 <G96.0 to G96.6>.

---

(8) 0.1% rapid traverse override select signal FHROV <G353.7>

[Classification] Input signal, path-by-path signal

---

(9) 0.1% rapid traverse override signal *FHRO0 to *FHRO9 <G352.0 to G352.7, G353.0 to G353.1>

[Classification] Input signal, path-by-path signal
[Function] An override can be applied in steps of 0.1%. This signal is used with the CNC as well.
[Operation] If bit 2 (OVE) of parameter No. 8001 is set to 0, the 0.1% rapid traverse override signals *FHRO0 to *FHRO9 <G352.0 to G352.7, G353.0 to G353.1> enable a rapid traverse override to be specified in steps of 0.1% when the 1% rapid traverse override select signal HROV <G96.7> and the 0.1% rapid traverse override select signal FHROV <G353.7> are set to “1”.

---

(10) Rapid traverse override signal ROV1 <G14.0>, ROV2 <G14.1>

[Classification] Input signal, path-by-path signal
[Function] Overrides the rapid traverse rate. Shared by the CNC.
[Operation] This signal is a two-bit code signal, and sets an override value as indicated Table 16.1.1 (aa).

Table 16.1.1 (aa)

Rapid traverse override signal		Override value
ROV2	ROV1	
0	0	100%
0	1	50%
1	0	25%
1	1	F0 (parameter No. 1421)

(11) First reference position return completion signals ZP1 to ZP8 <F94>
(12) Second reference position return completion signals ZP21 to ZP28 <F96>
(13) Third reference position return completion signals ZP31 to ZP38 <F98>
(14) Fourth reference position return completion signals ZP41 to ZP48 <F100>

[Classification] Output signal, axis-by-axis signal

[Function] Posts that the tool is at the reference position on a controlled axis. Shared by the CNC.

[Operation] The number at the end of a signal name indicates the number of a controlled axis.
Each of these signals is set to “1” when reference position return is completed, and the tool enters the in-position area.
Each of these signals is set to “0” when a movement is made from the reference position, an emergence stop occurs, or a servo alarm is issued.

(15) A/B phase detector disconnection alarm ignore signal NDCAL1 to NDCAL8 <Gn202>

[Classification] Input signal, axis-by-axis signal

[Function] Does not output the A/B phase detector hard disconnection alarm.

[Operation] While this signal is set to “1”, the A/B phase detector hard disconnection alarm is not output during speed specification based on PMC axis control. Fig. , "Timing chart of A/B phase detector disconnection alarm ignore signal" shows a timing chart.

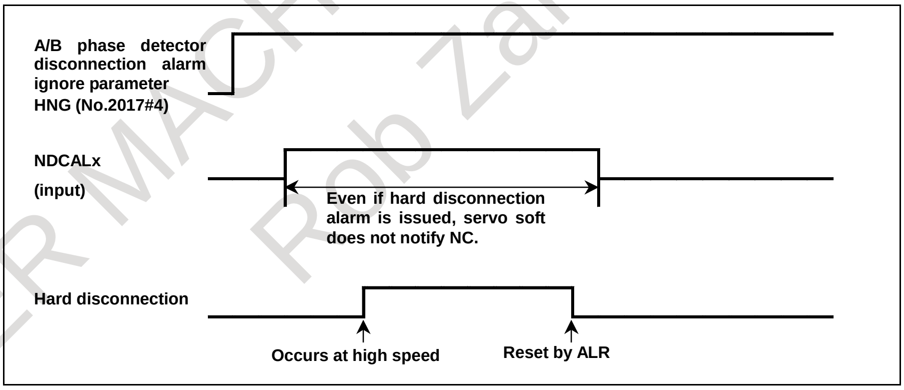


Fig. 16.1.1 (o) Timing chart of A/B phase detector disconnection alarm ignore signal

NOTE
1. Change the level of this signal from “1” to “0” before switching from a speed command to a position command.
2. After switching from a speed command to a position command, be sure to perform a manual reference position return operation before making a movement on the axis.
3. This function cannot be used when an absolute-position detector is attached.
4. To enable this signal, set bit 4 (HTN) of parameter No. 2017 to 1.

(16) Manual pulse magnification change signal HNDMP<Gn088.3>

[Classification] Input signal, path-by-path signal

[Function] In an external pulse synchronization function of the PMC axis control, the setting range of the weight of an external pulse (multiplier) is selected.
"±1/256 to ±127" or "±1/32 to ±1023" can be selected as a setting range.

This signal becomes effective for the following the axis control command signals (EC0g to EC6g <Gn143.0 to 6, Gn155.0 to 6, Gn167.0 to 6, Gn179.0 to 6>).

axis control command (Hexadecimal Code)	Operation
0Bh	External pulse synchronization - position coder
0Dh	External pulse synchronization - first manual handle
0Eh	External pulse synchronization - second manual handle
0Fh	External pulse synchronization - third manual handle

[Operation] In the Case of HNDMP<Gn088.3> is "0":

When setting a weight value, use the higher bits (EIF8g to EIF15g) to specify the integer part of an external pulse weight, and use the lower bits (EIF0g to EIF7g) to specify the fractional part.
The valid range is ±1/256 to ±127.

Integer part : Higher bits EIF8g to EIF15g (-127 to 127)

15	14	13	12	11	10	9	8
EIF15g	EIF14g	EIF13g	EIF12g	EIF11g	EIF10g	EIF9g	EIF8g

Fractional part : Lower bits EIF0g to EIF7g (0/256 to 255/256)

7	6	5	4	3	2	1	0
EIF7g	EIF6g	EIF5g	EIF4g	EIF3g	EIF2g	EIF1g	EIF0g

In the Case of HNDMP<Gn088.3> is "1":

When setting a weight value, use the higher bits (EIF5g to EIF15g) to specify the integer part of an external pulse weight, and use the lower bits (EIF0g to EIF4g) to specify the fractional part.
The valid range is ±1/32 to ±1023.

Integer part : Higher bits EIF5g to EIF15g (-1023 to 1023)

15	14	13	12	11	10	9	8	7	6	5
EIF15g	EIF14g	EIF13g	EIF12g	EIF11g	EIF10g	EIF9g	EIF8g	EIF7g	EIF6g	EIF5g

Fractional part : Lower bits EIF0g to EIF4g (0/32 to 31/32)

4	3	2	1	0
EIF4g	EIF3g	EIF2g	EIF1g	EIF0g

NOTE

The manual handle feed amount selection signals (MP1,MP2<Gn019.4,5>, MP21,MP22<Gn087.0,1>, MP31, MP32<Gn087.3,4>) do not influence external pulse synchronization weight.

Example of setting weight of pulse)

In the Case of HNDMP<Gn088.3> is “0”:

Setting of weight in fractional part

Set the lower bits (EIF0g to EIF7g) to specify the fractional part. (0/256 to 255/256)

Example) For weight 0.5

0.5 → 128/256 → “1000 0000”

“1000 0000” that converts 128 into a binary number is set to EIF0g-EIF7g.

Setting of negative weight

The two's complement expression is used for the setting of negative weight.

Example) For weight -2.5

2.5: “0000 0010 1000 0000”

↓ : The bit is reversed.

“1111 1101 0111 1111”

↓ : 1 is added to the 0th bit.

-2.5: “1111 1101 1000 0000”

List of example of setting weight of pulse

Axis control feedrate signal		Pulse weight
<EIF15g - EIF8g>	<EIF7g - EIF0g>	
00000000	00000000	× 0
00000000	00000001	× 1/256
00000000	00001000	× 1/32
00000001	00000000	× 1
00001010	00000000	× 10
01111111	00000000	× 127
11111111	11111111	× -1/256
11111111	11111000	× -1/32
11111111	00000000	× -1
11110110	00000000	× -10
10000001	00000000	× -127

In the Case of HNDMP<Gn088.3> is “1”:

Setting of weight in fractional part

Set the lower bits (EIF0g to EIF4g) to specify the fractional part. (0/32 to 31/32)

Example) For weight 0.5

0.5 → 16/32 → “1 0000”

“1 0000” that converts 16 into a binary number is set to EIF0g-EIF4g.

Setting of negative weight

The two's complement expression is used for the setting of negative weight.

Example) For weight -2.5

2.5: “0000 0000 0101 0000”

↓ : The bit is reversed.

“1111 1111 1010 1111”

↓ : 1 is added to the 0th bit.

-2.5: “1111 1111 1011 0000”

List of example of setting weight of pulse

Axis control feedrate signal		Pulse weight
<EIF15g – EIF5g>	<EIF4g - EIF0g>	
00000000000	00000	× 0
00000000000	00001	× 1/32
00000000001	00000	× 1
00000001010	00000	× 10
01111111111	00000	× 1023
11111111111	11111	× -1/32
11111111111	00000	× -1
11111110110	00000	× -10
10000000001	00000	× -1023

Signal address

- Signals for PMC axis control

DI → CNC

The signals below are direct signals for PMC axis control.
For the relationships with groups assigned to signals, see Table 16.1.1 (c), "Example of Path-by-path signal (PMC axis control override signal) allocation".

	#7	#6	#5	#4	#3	#2	#1	#0
X0004		ESKIP						
X0011		ESKIP#3						
X0013		ESKIP#2						

> **NOTE**
> The ESKIP signal is valid only for the axes assigned to paths 1 to 3.

PMC → CNC

Signals common to all groups (path-by-path signals)

The signals below are assigned on a path-by-path basis. For details of relationships with groups assigned to signals, see Table 16.1.1 (c).
Example:
EDRN#1 is valid for the axes assigned to groups 1 to 4.
ERT#3 is valid for the axes assigned to groups 9 to 12.

	#7	#6	#5	#4	#3	#2	#1	#0
G150	EDRN#1	ERT#1	EOVC#1				EROV2#1	EROV1#1
G151	*EFOV7#1 *EROV7#1	*EFOV6#1 *EROV6#1	*EFOV5#1 *EROV5#1	*EFOV4#1 *EROV4#1	*EFOV3#1 *EROV3#1	*EFOV2#1 *EROV2#1	*EFOV1#1 *EROV1#1	*EFOV0#1 *EROV0#1

	#7	#6	#5	#4	#3	#2	#1	#0
G1150	EDRN#2	ERT#2	EOVC#2				EROV2#2	EROV1#2
G1151	*EFOV7#2 *EROV7#2	*EFOV6#2 *EROV6#2	*EFOV5#2 *EROV5#2	*EFOV4#2 *EROV4#2	*EFOV3#2 *EROV3#2	*EFOV2#2 *EROV2#2	*EFOV1#2 *EROV1#2	*EFOV0#2 *EROV0#2

	#7	#6	#5	#4	#3	#2	#1	#0
G2150	EDRN#3	ERT#3	EOVC#3				EROV2#3	EROV1#3
G2151	*EFOV7#3 *EROV7#3	*EFOV6#3 *EROV6#3	*EFOV5#3 *EROV5#3	*EFOV4#3 *EROV4#3	*EFOV3#3 *EROV3#3	*EFOV2#3 *EROV2#3	*EFOV1#3 *EROV1#3	*EFOV0#3 *EROV0#3

	#7	#6	#5	#4	#3	#2	#1	#0
G3150	EDRN#4	ERT#4	EOVC#4				EROV2#4	EROV1#4
G3151	*EFOV7#4 *EROV7#4	*EFOV6#4 *EROV6#4	*EFOV5#4 *EROV5#4	*EFOV4#4 *EROV4#4	*EFOV3#4 *EROV3#4	*EFOV2#4 *EROV2#4	*EFOV1#4 *EROV1#4	*EFOV0#4 *EROV0#4

NOTE
Whether EOVC and *EFOV are group-by-group signals or path-by-path signals is determined by bit 5 (IFV) of parameter No. 8005.

Signals common to all groups (axis-by-axis signals)

The signals below are assigned on an axis-by-axis basis.

	#7	#6	#5	#4	#3	#2	#1	#0
G136	EAX8#1	EAX7#1	EAX6#1	EAX5#1	EAX4#1	EAX3#1	EAX2#1	EAX1#1
G1136	EAX8#2	EAX7#2	EAX6#2	EAX5#2	EAX4#2	EAX3#2	EAX2#2	EAX1#2
G2136	EAX8#3	EAX7#3	EAX6#3	EAX5#3	EAX4#3	EAX3#3	EAX2#3	EAX1#3
G3136	EAX8#4	EAX7#4	EAX6#4	EAX5#4	EAX4#4	EAX3#4	EAX2#4	EAX1#4

	#7	#6	#5	#4	#3	#2	#1	#0
G202	NDCAL8#1	NDCAL7#1	NDCAL6#1	NDCAL5#1	NDCAL4#1	NDCAL3#1	NDCAL2#1	NDCAL1#1
G1202	NDCAL8#2	NDCAL7#2	NDCAL6#2	NDCAL5#2	NDCAL4#2	NDCAL3#2	NDCAL2#2	NDCAL1#2
G2202	NDCAL8#3	NDCAL7#3	NDCAL6#3	NDCAL5#3	NDCAL4#3	NDCAL3#3	NDCAL2#3	NDCAL1#3
G3202	NDCAL8#4	NDCAL7#4	NDCAL6#4	NDCAL5#4	NDCAL4#4	NDCAL3#4	NDCAL2#4	NDCAL1#4

Group-by-group signals

The signals below are assigned on a group-by-group basis.

For group 1

	#7	#6	#5	#4	#3	#2	#1	#0
G142	EBUFA#1	ECLRA#1	ESTPA#1	ESOFA#1	ESBKA#1	EMBUFA#1	ELCKZA#1	EFINA#1
G143	EMSBKA#1	EC6A#1	EC5A#1	EC4A#1	EC3A#1	EC2A#1	EC1A#1	EC0A#1
G144	EIF7A#1	EIF6A#1	EIF5A#1	EIF4A#1	EIF3A#1	EIF2A#1	EIF1A#1	EIF0A#1
G145	EIF15A#1	EIF14A#1	EIF13A#1	EIF12A#1	EIF11A#1	EIF10A#1	EIF9A#1	EIF8A#1
G146	EID7A#1	EID6A#1	EID5A#1	EID4A#1	EID3A#1	EID2A#1	EID1A#1	EID0A#1
G147	EID15A#1	EID14A#1	EID13A#1	EID12A#1	EID11A#1	EID10A#1	EID9A#1	EID8A#1
G148	EID23A#1	EID22A#1	EID21A#1	EID20A#1	EID19A#1	EID18A#1	EID17A#1	EID16A#1
G149	EID31A#1	EID30A#1	EID29A#1	EID28A#1	EID27A#1	EID26A#1	EID25A#1	EID24A#1
G150			EOVCA#1					
G151	*EFOV7A#1 *EROV7A#1	*EFOV6A#1 *EROV6A#1	*EFOV5A#1 *EROV5A#1	*EFOV4A#1 *EROV4A#1	*EFOV3A#1 *EROV3A#1	*EFOV2A#1 *EROV2A#1	*EFOV1A#1 *EROV1A#1	*EFOV0A#1 *EROV0A#1

For group 2

	#7	#6	#5	#4	#3	#2	#1	#0
G154	EBUFB#1	ECLRB#1	ESTPB#1	ESOFB#1	ESBKB#1	EMBUFB#1	ELCKZB#1	EFINB#1
G155	EMSBKB#1	EC6B#1	EC5B#1	EC4B#1	EC3B#1	EC2B#1	EC1B#1	EC0B#1
G156	EIF7B#1	EIF6B#1	EIF5B#1	EIF4B#1	EIF3B#1	EIF2B#1	EIF1B#1	EIF0B#1
G157	EIF15B#1	EIF14B#1	EIF13B#1	EIF12B#1	EIF11B#1	EIF10B#1	EIF9B#1	EIF8B#1
G158	EID7B#1	EID6B#1	EID5B#1	EID4B#1	EID3B#1	EID2B#1	EID1B#1	EID0B#1
G159	EID15B#1	EID14B#1	EID13B#1	EID12B#1	EID11B#1	EID10B#1	EID9B#1	EID8B#1
G160	EID23B#1	EID22B#1	EID21B#1	EID20B#1	EID19B#1	EID18B#1	EID17B#1	EID16B#1
G161	EID31B#1	EID30B#1	EID29B#1	EID28B#1	EID27B#1	EID26B#1	EID25B#1	EID24B#1
G162			EOVCB#1					
G163	*EFOV7B#1 *EROV7B#1	*EFOV6B#1 *EROV6B#1	*EFOV5B#1 *EROV5B#1	*EFOV4B#1 *EROV4B#1	*EFOV3B#1 *EROV3B#1	*EFOV2B#1 *EROV2B#1	*EFOV1B#1 *EROV1B#1	*EFOV0B#1 *EROV0B#1

For group 3

	#7	#6	#5	#4	#3	#2	#1	#0
G166	EBUFC#1	ECLRC#1	ESTPC#1	ESOFC#1	ESBKC#1	EMBUFC#1	ELCKZC#1	EFINC#1
G167	EMSBKC#1	EC6C#1	EC5C#1	EC4C#1	EC3C#1	EC2C#1	EC1C#1	EC0C#1
G168	EIF7C#1	EIF6C#1	EIF5C#1	EIF4C#1	EIF3C#1	EIF2C#1	EIF1C#1	EIF0C#1
G169	EIF15C#1	EIF14C#1	EIF13C#1	EIF12C#1	EIF11C#1	EIF10C#1	EIF9C#1	EIF8C#1
G170	EID7C#1	EID6C#1	EID5C#1	EID4C#1	EID3C#1	EID2C#1	EID1C#1	EID0C#1
G171	EID15C#1	EID14C#1	EID13C#1	EID12C#1	EID11C#1	EID10C#1	EID9C#1	EID8C#1
G172	EID23C#1	EID22C#1	EID21C#1	EID20C#1	EID19C#1	EID18C#1	EID17C#1	EID16C#1
G173	EID31C#1	EID30C#1	EID29C#1	EID28C#1	EID27C#1	EID26C#1	EID25C#1	EID24C#1
G174			EOVCC#1					
G175	*EFOV7C#1 *EROV7C#1	*EFOV6C#1 *EROV6C#1	*EFOV5C#1 *EROV5C#1	*EFOV4C#1 *EROV4C#1	*EFOV3C#1 *EROV3C#1	*EFOV2C#1 *EROV2C#1	*EFOV1C#1 *EROV1C#1	*EFOV0C#1 *EROV0C#1

For group 4

	#7	#6	#5	#4	#3	#2	#1	#0
G178	EBUFD#1	ECLRD#1	ESTPD#1	ESOFD#1	ESBKD#1	EMBUFD#1	ELCKZD#1	EFIND#1
G179	EMSBKD#1	EC6D#1	EC5D#1	EC4D#1	EC3D#1	EC2D#1	EC1D#1	EC0D#1
G180	EIF7D#1	EIF6D#1	EIF5D#1	EIF4D#1	EIF3D#1	EIF2D#1	EIF1D#1	EIF0D#1
G181	EIF15D#1	EIF14D#1	EIF13D#1	EIF12D#1	EIF11D#1	EIF10D#1	EIF9D#1	EIF8D#1
G182	EID7D#1	EID6D#1	EID5D#1	EID4D#1	EID3D#1	EID2D#1	EID1D#1	EID0D#1
G183	EID15D#1	EID14D#1	EID13D#1	EID12D#1	EID11D#1	EID10D#1	EID9D#1	EID8D#1
G184	EID23D#1	EID22D#1	EID21D#1	EID20D#1	EID19D#1	EID18D#1	EID17D#1	EID16D#1
G185	EID31D#1	EID30D#1	EID29D#1	EID28D#1	EID27D#1	EID26D#1	EID25D#1	EID24D#1
G186			EOVCD#1					
G187	*EFOV7D#1 *EROV7D#1	*EFOV6D#1 *EROV6D#1	*EFOV5D#1 *EROV5D#1	*EFOV4D#1 *EROV4D#1	*EFOV3D#1 *EROV3D#1	*EFOV2D#1 *EROV2D#1	*EFOV1D#1 *EROV1D#1	*EFOV0D#1 *EROV0D#1

For group 5

Address	#7	#6	#5	#4	#3	#2	#1	#0
G1142	EBUFA#2	ECLRA#2	ESTPA#2	ESOFA#2	ESBKA#2	EMBUFA#2	ELCKZA#2	EFINA#2
G1143	EMSBKA#2	EC6A#2	EC5A#2	EC4A#2	EC3A#2	EC2A#2	EC1A#2	EC0A#2
G1144	EIF7A#2	EIF6A#2	EIF5A#2	EIF4A#2	EIF3A#2	EIF2A#2	EIF1A#2	EIF0A#2
G1145	EIF15A#2	EIF14A#2	EIF13A#2	EIF12A#2	EIF11A#2	EIF10A#2	EIF9A#2	EIF8A#2
G1146	EID7A#2	EID6A#2	EID5A#2	EID4A#2	EID3A#2	EID2A#2	EID1A#2	EID0A#2
G1147	EID15A#2	EID14A#2	EID13A#2	EID12A#2	EID11A#2	EID10A#2	EID9A#2	EID8A#2
G1148	EID23A#2	EID22A#2	EID21A#2	EID20A#2	EID19A#2	EID18A#2	EID17A#2	EID16A#2
G1149	EID31A#2	EID30A#2	EID29A#2	EID28A#2	EID27A#2	EID26A#2	EID25A#2	EID24A#2
G1150			EOVCA#2					
G1151	*EFOV7A#2 *EROV7A#2	*EFOV6A#2 *EROV6A#2	*EFOV5A#2 *EROV5A#2	*EFOV4A#2 *EROV4A#2	*EFOV3A#2 *EROV3A#2	*EFOV2A#2 *EROV2A#2	*EFOV1A#2 *EROV1A#2	*EFOV0A#2 *EROV0A#2

For group 6

Address	#7	#6	#5	#4	#3	#2	#1	#0
G1154	EBUFB#2	ECLRB#2	ESTPB#2	ESOFB#2	ESBKB#2	EMBUFB#2	ELCKZB#2	EFINB#2
G1155	EMSBKB#2	EC6B#2	EC5B#2	EC4B#2	EC3B#2	EC2B#2	EC1B#2	EC0B#2
G1156	EIF7B#2	EIF6B#2	EIF5B#2	EIF4B#2	EIF3B#2	EIF2B#2	EIF1B#2	EIF0B#2
G1157	EIF15B#2	EIF14B#2	EIF13B#2	EIF12B#2	EIF11B#2	EIF10B#2	EIF9B#2	EIF8B#2
G1158	EID7B#2	EID6B#2	EID5B#2	EID4B#2	EID3B#2	EID2B#2	EID1B#2	EID0B#2
G1159	EID15B#2	EID14B#2	EID13B#2	EID12B#2	EID11B#2	EID10B#2	EID9B#2	EID8B#2
G1160	EID23B#2	EID22B#2	EID21B#2	EID20B#2	EID19B#2	EID18B#2	EID17B#2	EID16B#2
G1161	EID31B#2	EID30B#2	EID29B#2	EID28B#2	EID27B#2	EID26B#2	EID25B#2	EID24B#2
G1162			EOVCB#2					
G1163	*EFOV7B#2 *EROV7B#2	*EFOV6B#2 *EROV6B#2	*EFOV5B#2 *EROV5B#2	*EFOV4B#2 *EROV4B#2	*EFOV3B#2 *EROV3B#2	*EFOV2B#2 *EROV2B#2	*EFOV1B#2 *EROV1B#2	*EFOV0B#2 *EROV0B#2

For group 7

Address	#7	#6	#5	#4	#3	#2	#1	#0
G1166	EBUFC#2	ECLRC#2	ESTPC#2	ESOFC#2	ESBKC#2	EMBUFC#2	ELCKZC#2	EFINC#2
G1167	EMSBKC#2	EC6C#2	EC5C#2	EC4C#2	EC3C#2	EC2C#2	EC1C#2	EC0C#2
G1168	EIF7C#2	EIF6C#2	EIF5C#2	EIF4C#2	EIF3C#2	EIF2C#2	EIF1C#2	EIF0C#2
G1169	EIF15C#2	EIF14C#2	EIF13C#2	EIF12C#2	EIF11C#2	EIF10C#2	EIF9C#2	EIF8C#2
G1170	EID7C#2	EID6C#2	EID5C#2	EID4C#2	EID3C#2	EID2C#2	EID1C#2	EID0C#2
G1171	EID15C#2	EID14C#2	EID13C#2	EID12C#2	EID11C#2	EID10C#2	EID9C#2	EID8C#2
G1172	EID23C#2	EID22C#2	EID21C#2	EID20C#2	EID19C#2	EID18C#2	EID17C#2	EID16C#2
G1173	EID31C#2	EID30C#2	EID29C#2	EID28C#2	EID27C#2	EID26C#2	EID25C#2	EID24C#2
G1174			EOVCC#2					
G1175	*EFOV7C#2 *EROV7C#2	*EFOV6C#2 *EROV6C#2	*EFOV5C#2 *EROV5C#2	*EFOV4C#2 *EROV4C#2	*EFOV3C#2 *EROV3C#2	*EFOV2C#2 *EROV2C#2	*EFOV1C#2 *EROV1C#2	*EFOV0C#2 *EROV0C#2

For group 8

Address	#7	#6	#5	#4	#3	#2	#1	#0
G1178	EBUFD#2	ECLRD#2	ESTPD#2	ESOFD#2	ESBKD#2	EMBUFD#2	ELCKZD#2	EFIND#2
G1179	EMSBKD#2	EC6D#2	EC5D#2	EC4D#2	EC3D#2	EC2D#2	EC1D#2	EC0D#2
G1180	EIF7D#2	EIF6D#2	EIF5D#2	EIF4D#2	EIF3D#2	EIF2D#2	EIF1D#2	EIF0D#2
G1181	EIF15D#2	EIF14D#2	EIF13D#2	EIF12D#2	EIF11D#2	EIF10D#2	EIF9D#2	EIF8D#2
G1182	EID7D#2	EID6D#2	EID5D#2	EID4D#2	EID3D#2	EID2D#2	EID1D#2	EID0D#2
G1183	EID15D#2	EID14D#2	EID13D#2	EID12D#2	EID11D#2	EID10D#2	EID9D#2	EID8D#2
G1184	EID23D#2	EID22D#2	EID21D#2	EID20D#2	EID19D#2	EID18D#2	EID17D#2	EID16D#2
G1185	EID31D#2	EID30D#2	EID29D#2	EID28D#2	EID27D#2	EID26D#2	EID25D#2	EID24D#2
G1186			EOVCD#2					
G1187	*EFOV7D#2 *EROV7D#2	*EFOV6D#2 *EROV6D#2	*EFOV5D#2 *EROV5D#2	*EFOV4D#2 *EROV4D#2	*EFOV3D#2 *EROV3D#2	*EFOV2D#2 *EROV2D#2	*EFOV1D#2 *EROV1D#2	*EFOV0D#2 *EROV0D#2

For group 9

	#7	#6	#5	#4	#3	#2	#1	#0
G2142	EBUFA#3	ECLRA#3	ESTPA#3	ESOFA#3	ESBKA#3	EMBUFA#3	ELCKZA#3	EFINA#3
G2143	EMSBKA#3	EC6A#3	EC5A#3	EC4A#3	EC3A#3	EC2A#3	EC1A#3	EC0A#3
G2144	EIF7A#3	EIF6A#3	EIF5A#3	EIF4A#3	EIF3A#3	EIF2A#3	EIF1A#3	EIF0A#3
G2145	EIF15A#3	EIF14A#3	EIF13A#3	EIF12A#3	EIF11A#3	EIF10A#3	EIF9A#3	EIF8A#3
G2146	EID7A#3	EID6A#3	EID5A#3	EID4A#3	EID3A#3	EID2A#3	EID1A#3	EID0A#3
G2147	EID15A#3	EID14A#3	EID13A#3	EID12A#3	EID11A#3	EID10A#3	EID9A#3	EID8A#3
G2148	EID23A#3	EID22A#3	EID21A#3	EID20A#3	EID19A#3	EID18A#3	EID17A#3	EID16A#3
G2149	EID31A#3	EID30A#3	EID29A#3	EID28A#3	EID27A#3	EID26A#3	EID25A#3	EID24A#3
G2150			EOVCA#3					
G2151	*EFOV7A#3 *EROV7A#3	*EFOV6A#3 *EROV6A#3	*EFOV5A#3 *EROV5A#3	*EFOV4A#3 *EROV4A#3	*EFOV3A#3 *EROV3A#3	*EFOV2A#3 *EROV2A#3	*EFOV1A#3 *EROV1A#3	*EFOV0A#3 *EROV0A#3

For group 10

	#7	#6	#5	#4	#3	#2	#1	#0
G2154	EBUFB#3	ECLRB#3	ESTPB#3	ESOFB#3	ESBKB#3	EMBUFB#3	ELCKZB#3	EFINB#3
G2155	EMSBKB#3	EC6B#3	EC5B#3	EC4B#3	EC3B#3	EC2B#3	EC1B#3	EC0B#3
G2156	EIF7B#3	EIF6B#3	EIF5B#3	EIF4B#3	EIF3B#3	EIF2B#3	EIF1B#3	EIF0B#3
G2157	EIF15B#3	EIF14B#3	EIF13B#3	EIF12B#3	EIF11B#3	EIF10B#3	EIF9B#3	EIF8B#3
G2158	EID7B#3	EID6B#3	EID5B#3	EID4B#3	EID3B#3	EID2B#3	EID1B#3	EID0B#3
G2159	EID15B#3	EID14B#3	EID13B#3	EID12B#3	EID11B#3	EID10B#3	EID9B#3	EID8B#3
G2160	EID23B#3	EID22B#3	EID21B#3	EID20B#3	EID19B#3	EID18B#3	EID17B#3	EID16B#3
G2161	EID31B#3	EID30B#3	EID29B#3	EID28B#3	EID27B#3	EID26B#3	EID25B#3	EID24B#3
G2162			EOVCB#3					
G2163	*EFOV7B#3 *EROV7B#3	*EFOV6B#3 *EROV6B#3	*EFOV5B#3 *EROV5B#3	*EFOV4B#3 *EROV4B#3	*EFOV3B#3 *EROV3B#3	*EFOV2B#3 *EROV2B#3	*EFOV1B#3 *EROV1B#3	*EFOV0B#3 *EROV0B#3

For group 11

	#7	#6	#5	#4	#3	#2	#1	#0
G2166	EBUFC#3	ECLRC#3	ESTPC#3	ESOFC#3	ESBKC#3	EMBUFC#3	ELCKZC#3	EFINC#3
G2167	EMSBKC#3	EC6C#3	EC5C#3	EC4C#3	EC3C#3	EC2C#3	EC1C#3	EC0C#3
G2168	EIF7C#3	EIF6C#3	EIF5C#3	EIF4C#3	EIF3C#3	EIF2C#3	EIF1C#3	EIF0C#3
G2169	EIF15C#3	EIF14C#3	EIF13C#3	EIF12C#3	EIF11C#3	EIF10C#3	EIF9C#3	EIF8C#3
G2170	EID7C#3	EID6C#3	EID5C#3	EID4C#3	EID3C#3	EID2C#3	EID1C#3	EID0C#3
G2171	EID15C#3	EID14C#3	EID13C#3	EID12C#3	EID11C#3	EID10C#3	EID9C#3	EID8C#3
G2172	EID23C#3	EID22C#3	EID21C#3	EID20C#3	EID19C#3	EID18C#3	EID17C#3	EID16C#3
G2173	EID31C#3	EID30C#3	EID29C#3	EID28C#3	EID27C#3	EID26C#3	EID25C#3	EID24C#3
G2174			EOVCC#3					
G2175	*EFOV7C#3 *EROV7C#3	*EFOV6C#3 *EROV6C#3	*EFOV5C#3 *EROV5C#3	*EFOV4C#3 *EROV4C#3	*EFOV3C#3 *EROV3C#3	*EFOV2C#3 *EROV2C#3	*EFOV1C#3 *EROV1C#3	*EFOV0C#3 *EROV0C#3

For group 12

	#7	#6	#5	#4	#3	#2	#1	#0
G2178	EBUFD#3	ECLRD#3	ESTPD#3	ESOFD#3	ESBKD#3	EMBUFD#3	ELCKZD#3	EFIND#3
G2179	EMSBKD#3	EC6D#3	EC5D#3	EC4D#3	EC3D#3	EC2D#3	EC1D#3	EC0D#3
G2180	EIF7D#3	EIF6D#3	EIF5D#3	EIF4D#3	EIF3D#3	EIF2D#3	EIF1D#3	EIF0D#3
G2181	EIF15D#3	EIF14D#3	EIF13D#3	EIF12D#3	EIF11D#3	EIF10D#3	EIF9D#3	EIF8D#3
G2182	EID7D#3	EID6D#3	EID5D#3	EID4D#3	EID3D#3	EID2D#3	EID1D#3	EID0D#3
G2183	EID15D#3	EID14D#3	EID13D#3	EID12D#3	EID11D#3	EID10D#3	EID9D#3	EID8D#3
G2184	EID23D#3	EID22D#3	EID21D#3	EID20D#3	EID19D#3	EID18D#3	EID17D#3	EID16D#3
G2185	EID31D#3	EID30D#3	EID29D#3	EID28D#3	EID27D#3	EID26D#3	EID25D#3	EID24D#3
G2186			EOVCD#3					
G2187	*EFOV7D#3 *EROV7D#3	*EFOV6D#3 *EROV6D#3	*EFOV5D#3 *EROV5D#3	*EFOV4D#3 *EROV4D#3	*EFOV3D#3 *EROV3D#3	*EFOV2D#3 *EROV2D#3	*EFOV1D#3 *EROV1D#3	*EFOV0D#3 *EROV0D#3

For group 13

	#7	#6	#5	#4	#3	#2	#1	#0
G3142	EBUFA#4	ECLRA#4	ESTPA#4	ESOFA#4	ESBKA#4	EMBUFA#4	ELCKZA#4	EFINA#4
G3143	EMSBKA#4	EC6A#4	EC5A#4	EC4A#4	EC3A#4	EC2A#4	EC1A#4	EC0A#4
G3144	EIF7A#4	EIF6A#4	EIF5A#4	EIF4A#4	EIF3A#4	EIF2A#4	EIF1A#4	EIF0A#4
G3145	EIF15A#4	EIF14A#4	EIF13A#4	EIF12A#4	EIF11A#4	EIF10A#4	EIF9A#4	EIF8A#4
G3146	EID7A#4	EID6A#4	EID5A#4	EID4A#4	EID3A#4	EID2A#4	EID1A#4	EID0A#4
G3147	EID15A#4	EID14A#4	EID13A#4	EID12A#4	EID11A#4	EID10A#4	EID9A#4	EID8A#4
G3148	EID23A#4	EID22A#4	EID21A#4	EID20A#4	EID19A#4	EID18A#4	EID17A#4	EID16A#4
G3149	EID31A#4	EID30A#4	EID29A#4	EID28A#4	EID27A#4	EID26A#4	EID25A#4	EID24A#4
G3150			EOVCA#4					
G3151	*EFOV7A#4 *EROV7A#4	*EFOV6A#4 *EROV6A#4	*EFOV5A#4 *EROV5A#4	*EFOV4A#4 *EROV4A#4	*EFOV3A#4 *EROV3A#4	*EFOV2A#4 *EROV2A#4	*EFOV1A#4 *EROV1A#4	*EFOV0A#4 *EROV0A#4

For group 14

	#7	#6	#5	#4	#3	#2	#1	#0
G3154	EBUFB#4	ECLRB#4	ESTPB#4	ESOFB#4	ESBKB#4	EMBUFB#4	ELCKZB#4	EFINB#4
G3155	EMSBKB#4	EC6B#4	EC5B#4	EC4B#4	EC3B#4	EC2B#4	EC1B#4	EC0B#4
G3156	EIF7B#4	EIF6B#4	EIF5B#4	EIF4B#4	EIF3B#4	EIF2B#4	EIF1B#4	EIF0B#4
G3157	EIF15B#4	EIF14B#4	EIF13B#4	EIF12B#4	EIF11B#4	EIF10B#4	EIF9B#4	EIF8B#4
G3158	EID7B#4	EID6B#4	EID5B#4	EID4B#4	EID3B#4	EID2B#4	EID1B#4	EID0B#4
G3159	EID15B#4	EID14B#4	EID13B#4	EID12B#4	EID11B#4	EID10B#4	EID9B#4	EID8B#4
G3160	EID23B#4	EID22B#4	EID21B#4	EID20B#4	EID19B#4	EID18B#4	EID17B#4	EID16B#4
G3161	EID31B#4	EID30B#4	EID29B#4	EID28B#4	EID27B#4	EID26B#4	EID25B#4	EID24B#4
G3162			EOVCB#4					
G3163	*EFOV7B#4 *EROV7B#4	*EFOV6B#4 *EROV6B#4	*EFOV5B#4 *EROV5B#4	*EFOV4B#4 *EROV4B#4	*EFOV3B#4 *EROV3B#4	*EFOV2B#4 *EROV2B#4	*EFOV1B#4 *EROV1B#4	*EFOV0B#4 *EROV0B#4

For group 15

Address	#7	#6	#5	#4	#3	#2	#1	#0
G3166	EBUFC#4	ECLRC#4	ESTPC#4	ESOFC#4	ESBKC#4	EMBUFC#4	ELCKZC#4	EFINC#4
G3167	EMSBKC#4	EC6C#4	EC5C#4	EC4C#4	EC3C#4	EC2C#4	EC1C#4	EC0C#4
G3168	EIF7C#4	EIF6C#4	EIF5C#4	EIF4C#4	EIF3C#4	EIF2C#4	EIF1C#4	EIF0C#4
G3169	EIF15C#4	EIF14C#4	EIF13C#4	EIF12C#4	EIF11C#4	EIF10C#4	EIF9C#4	EIF8C#4
G3170	EID7C#4	EID6C#4	EID5C#4	EID4C#4	EID3C#4	EID2C#4	EID1C#4	EID0C#4
G3171	EID15C#4	EID14C#4	EID13C#4	EID12C#4	EID11C#4	EID10C#4	EID9C#4	EID8C#4
G3172	EID23C#4	EID22C#4	EID21C#4	EID20C#4	EID19C#4	EID18C#4	EID17C#4	EID16C#4
G3173	EID31C#4	EID30C#4	EID29C#4	EID28C#4	EID27C#4	EID26C#4	EID25C#4	EID24C#4
G3174			EOVCC#4					
G3175	*EFOV7C#4 *EROV7C#4	*EFOV6C#4 *EROV6C#4	*EFOV5C#4 *EROV5C#4	*EFOV4C#4 *EROV4C#4	*EFOV3C#4 *EROV3C#4	*EFOV2C#4 *EROV2C#4	*EFOV1C#4 *EROV1C#4	*EFOV0C#4 *EROV0C#4

For group 16

Address	#7	#6	#5	#4	#3	#2	#1	#0
G3178	EBUFD#4	ECLRD#4	ESTPD#4	ESOFD#4	ESBKD#4	EMBUFD#4	ELCKZD#4	EFIND#4
G3179	EMSBKD#4	EC6D#4	EC5D#4	EC4D#4	EC3D#4	EC2D#4	EC1D#4	EC0D#4
G3180	EIF7D#4	EIF6D#4	EIF5D#4	EIF4D#4	EIF3D#4	EIF2D#4	EIF1D#4	EIF0D#4
G3181	EIF15D#4	EIF14D#4	EIF13D#4	EIF12D#4	EIF11D#4	EIF10D#4	EIF9D#4	EIF8D#4
G3182	EID7D#4	EID6D#4	EID5D#4	EID4D#4	EID3D#4	EID2D#4	EID1D#4	EID0D#4
G3183	EID15D#4	EID14D#4	EID13D#4	EID12D#4	EID11D#4	EID10D#4	EID9D#4	EID8D#4
G3184	EID23D#4	EID22D#4	EID21D#4	EID20D#4	EID19D#4	EID18D#4	EID17D#4	EID16D#4
G3185	EID31D#4	EID30D#4	EID29D#4	EID28D#4	EID27D#4	EID26D#4	EID25D#4	EID24D#4
G3186			EOVCD#4					
G3187	*EFOV7D#4 *EROV7D#4	*EFOV6D#4 *EROV6D#4	*EFOV5D#4 *EROV5D#4	*EFOV4D#4 *EROV4D#4	*EFOV3D#4 *EROV3D#4	*EFOV2D#4 *EROV2D#4	*EFOV1D#4 *EROV1D#4	*EFOV0D#4 *EROV0D#4

NOTE
1. The bits other than the override cancellation signal EOVC at the address where EOVC is positioned are path-by-path signals. In the case of group 1, for example, the bits <G150.0 to 4,6,7> other than G150.5 are path-by-path signals.
2. Whether EOVC and *EFOV are group-by-group signals or path-by-path signals is determined by bit 5 (IFV) of parameter No. 8005.

CNC→PMC

Signals common to all groups (path-by-path signals)

	#7	#6	#5	#4	#3	#2	#1	#0
F129	*EAXSL		EOV0					

Signals common to all groups (axis-by-axis signals)

	#7	#6	#5	#4	#3	#2	#1	#0
F112	EADEN8#1	EADEN7#1	EADEN6#1	EADEN5#1	EADEN4#1	EADEN3#1	EADEN2#1	EADEN1#1
F182	EACNT8#1	EACNT7#1	EACNT6#1	EACNT5#1	EACNT4#1	EACNT3#1	EACNT2#1	EACNT1#1
F190	TRQM8#1	TRQM7#1	TRQM6#1	TRQM5#1	TRQM4#1	TRQM3#1	TRQM2#1	TRQM1#1

Group-by-group signals

For group 1

	#7	#6	#5	#4	#3	#2	#1	#0
F130	EBSYA#1	EOTNA#1	EOTPA#1	EGENA#1	EDENA#1	EIALA#1	ECKZA#1	EINPA#1
F131					EMF3A#1	EMF2A#1	EABUFA#1	EMFA#1
F132	EM28A#1	EM24A#1	EM22A#1	EM21A#1	EM18A#1	EM14A#1	EM12A#1	EM11A#1
F142	EM48A#1	EM44A#1	EM42A#1	EM41A#1	EM38A#1	EM34A#1	EM32A#1	EM31A#1

For group 2

	#7	#6	#5	#4	#3	#2	#1	#0
F133	EBSYB#1	EOTNB#1	EOTPB#1	EGENB#1	EDENB#1	EIALB#1	ECKZB#1	EINPB#1
F134					EMF3B#1	EMF2B#1	EABUFB#1	EMFB#1
F135	EM28B#1	EM24B#1	EM22B#1	EM21B#1	EM18B#1	EM14B#1	EM12B#1	EM11B#1
F145	EM48B#1	EM44B#1	EM42B#1	EM41B#1	EM38B#1	EM34B#1	EM32B#1	EM31B#1

For group 3

	#7	#6	#5	#4	#3	#2	#1	#0
F136	EBSYC#1	EOTNC#1	EOTPC#1	EGENC#1	EDENC#1	EIALC#1	ECKZC#1	EINPC#1
F137					EMF3C#1	EMF2C#1	EABUFC#1	EMFC#1
F138	EM28C#1	EM24C#1	EM22C#1	EM21C#1	EM18C#1	EM14C#1	EM12C#1	EM11C#1
F148	EM48C#1	EM44C#1	EM42C#1	EM41C#1	EM38C#1	EM34C#1	EM32C#1	EM31C#1

For group 4

	#7	#6	#5	#4	#3	#2	#1	#0
F139	EBSYD#1	EOTND#1	EOTPD#1	EGEND#1	EDEND#1	EIALD#1	ECKZD#1	EINPD#1
F140					EMF3D#1	EMF2D#1	EABUFD#1	EMFD#1
F141	EM28D#1	EM24D#1	EM22D#1	EM21D#1	EM18D#1	EM14D#1	EM12D#1	EM11D#1
F151	EM48D#1	EM44D#1	EM42D#1	EM41D#1	EM38D#1	EM34D#1	EM32D#1	EM31D#1

For group 5

	#7	#6	#5	#4	#3	#2	#1	#0
F1130	EBSYA#2	EOTNA#2	EOTPA#2	EGENA#2	EDENA#2	EIALA#2	ECKZA#2	EINPA#2
F1131					EMF3A#2	EMF2A#2	EABUFA#2	EMFA#2
F1132	EM28A#2	EM24A#2	EM22A#2	EM21A#2	EM18A#2	EM14A#2	EM12A#2	EM11A#2
F1142	EM48A#2	EM44A#2	EM42A#2	EM41A#2	EM38A#2	EM34A#2	EM32A#2	EM31A#2

For group 6

	#7	#6	#5	#4	#3	#2	#1	#0
F1133	EBSYB#2	EOTNB#2	EOTPB#2	EGENB#2	EDENB#2	EIALB#2	ECKZB#2	EINPB#2
F1134					EMF3B#2	EMF2B#2	EABUFB#2	EMFB#2
F1135	EM28B#2	EM24B#2	EM22B#2	EM21B#2	EM18B#2	EM14B#2	EM12B#2	EM11B#2
F1145	EM48B#2	EM44B#2	EM42B#2	EM41B#2	EM38B#2	EM34B#2	EM32B#2	EM31B#2

For group 7

	#7	#6	#5	#4	#3	#2	#1	#0
F1136	EBSYC#2	EOTNC#2	EOTPC#2	EGENC#2	EDENC#2	EIALC#2	ECKZC#2	EINPC#2
F1137					EMF3C#2	EMF2C#2	EABUFC#2	EMFC#2
F1138	EM28C#2	EM24C#2	EM22C#2	EM21C#2	EM18C#2	EM14C#2	EM12C#2	EM11C#2
F1148	EM48C#2	EM44C#2	EM42C#2	EM41C#2	EM38C#2	EM34C#2	EM32C#2	EM31C#2

For group 8

	#7	#6	#5	#4	#3	#2	#1	#0
F1139	EBSYD#2	EOTND#2	EOTPD#2	EGEND#2	EDEND#2	EIALD#2	ECKZD#2	EINPD#2
F1140					EMF3D#2	EMF2D#2	EABUFD#2	EMFD#2
F1141	EM28D#2	EM24D#2	EM22D#2	EM21D#2	EM18D#2	EM14D#2	EM12D#2	EM11D#2
F1151	EM48D#2	EM44D#2	EM42D#2	EM41D#2	EM38D#2	EM34D#2	EM32D#2	EM31D#2

For group 9

	#7	#6	#5	#4	#3	#2	#1	#0
F2130	EBSYA#3	EOTNA#3	EOTPA#3	EGENA#3	EDENA#3	EIALA#3	ECKZA#3	EINPA#3
F2131					EMF3A#3	EMF2A#3	EABUFA#3	EMFA#3
F2132	EM28A#3	EM24A#3	EM22A#3	EM21A#3	EM18A#3	EM14A#3	EM12A#3	EM11A#3
F2142	EM48A#3	EM44A#3	EM42A#3	EM41A#3	EM38A#3	EM34A#3	EM32A#3	EM31A#3

For group 10

	#7	#6	#5	#4	#3	#2	#1	#0
F2133	EBSYB#3	EOTNB#3	EOTPB#3	EGENB#3	EDENB#3	EIALB#3	ECKZB#3	EINPB#3
F2134					EMF3B#3	EMF2B#3	EABUFB#3	EMFB#3
F2135	EM28B#3	EM24B#3	EM22B#3	EM21B#3	EM18B#3	EM14B#3	EM12B#3	EM11B#3
F2145	EM48B#3	EM44B#3	EM42B#3	EM41B#3	EM38B#3	EM34B#3	EM32B#3	EM31B#3

For group 11

	#7	#6	#5	#4	#3	#2	#1	#0
F2136	EBSYC#3	EOTNC#3	EOTPC#3	EGENC#3	EDENC#3	EIALC#3	ECKZC#3	EINPC#3
F2137					EMF3C#3	EMF2C#3	EABUFC#3	EMFC#3
F2138	EM28C#3	EM24C#3	EM22C#3	EM21C#3	EM18C#3	EM14C#3	EM12C#3	EM11C#3
F2148	EM48C#3	EM44C#3	EM42C#3	EM41C#3	EM38C#3	EM34C#3	EM32C#3	EM31C#3

For group 12

	#7	#6	#5	#4	#3	#2	#1	#0
F2139	EBSYD#3	EOTND#3	EOTPD#3	EGEND#3	EDEND#3	EIALD#3	ECKZD#3	EINPD#3
F2140					EMF3D#3	EMF2D#3	EABUFD#3	EMFD#3
F2141	EM28D#3	EM24D#3	EM22D#3	EM21D#3	EM18D#3	EM14D#3	EM12D#3	EM11D#3
F2151	EM48D#3	EM44D#3	EM42D#3	EM41D#3	EM38D#3	EM34D#3	EM32D#3	EM31D#3

For group 13

	#7	#6	#5	#4	#3	#2	#1	#0
F3130	EBSYA#4	EOTNA#4	EOTPA#4	EGENA#4	EDENA#4	EIALA#4	ECKZA#4	EINPA#4
F3131					EMF3A#4	EMF2A#4	EABUFA#4	EMFA#4
F3132	EM28A#4	EM24A#4	EM22A#4	EM21A#4	EM18A#4	EM14A#4	EM12A#4	EM11A#4
F3142	EM48A#4	EM44A#4	EM42A#4	EM41A#4	EM38A#4	EM34A#4	EM32A#4	EM31A#4

For group 14

	#7	#6	#5	#4	#3	#2	#1	#0
F3133	EBSYB#4	EOTNB#4	EOTPB#4	EGENB#4	EDENB#4	EIALB#4	ECKZB#4	EINPB#4
F3134					EMF3B#4	EMF2B#4	EABUFB#4	EMFB#4
F3135	EM28B#4	EM24B#4	EM22B#4	EM21B#4	EM18B#4	EM14B#4	EM12B#4	EM11B#4
F3145	EM48B#4	EM44B#4	EM42B#4	EM41B#4	EM38B#4	EM34B#4	EM32B#4	EM31B#4

For group 15

	#7	#6	#5	#4	#3	#2	#1	#0
F3136	EBSYC#4	EOTNC#4	EOTPC#4	EGENC#4	EDENC#4	EIALC#4	ECKZC#4	EINPC#4
F3137					EMF3C#4	EMF2C#4	EABUFC#4	EMFC#4
F3138	EM28C#4	EM24C#4	EM22C#4	EM21C#4	EM18C#4	EM14C#4	EM12C#4	EM11C#4
F3148	EM48C#4	EM44C#4	EM42C#4	EM41C#4	EM38C#4	EM34C#4	EM32C#4	EM31C#4

For group 16

	#7	#6	#5	#4	#3	#2	#1	#0
F3139	EBSYD#4	EOTND#4	EOTPD#4	EGEND#4	EDEND#4	EIALD#4	ECKZD#4	EINPD#4
F3140					EMF3D#4	EMF2D#4	EABUFD#4	EMFD#4
F3141	EM28D#4	EM24D#4	EM22D#4	EM21D#4	EM18D#4	EM14D#4	EM12D#4	EM11D#4
F3151	EM48D#4	EM44D#4	EM42D#4	EM41D#4	EM38D#4	EM34D#4	EM32D#4	EM31D#4

Related signals

DI → CNC

The signals below are direct signals related to PMC axis control.

Address	#7	#6	#5	#4	#3	#2	#1	#0
X004	SKIP							
X011	SKIP[#3]							
X013	SKIP[#2]							

PMC → CNC

The signals below are input signals related to PMC axis control.
For axis-type signals, addresses for the 1st axis to 8th axis are indicated. For path-type signals, addresses for group 1 are indicated.

Address	#7	#6	#5	#4	#3	#2	#1	#0
G014							ROV2	ROV1
G088					HNDMP			
G096	HROV	*HROV6	*HROV5	*HROV4	*HROV3	*HROV2	*HROV1	*HROV0
G101	*+ED28	*+ED27	*+ED26	*+ED25	*+ED24	*+ED23	*+ED22	*+ED21
G103	*-ED28	*-ED27	*-ED26	*-ED25	*-ED24	*-ED23	*-ED22	*-ED21
G106	MI8	MI7	MI6	MI5	MI4	MI3	MI2	MI1
G107	*+ED38	*+ED37	*+ED36	*+ED35	*+ED34	*+ED33	*+ED32	*+ED31
G109	*-ED38	*-ED37	*-ED36	*-ED35	*-ED34	*-ED33	*-ED32	*-ED31
G118	*+ED8	*+ED7	*+ED6	*+ED5	*+ED4	*+ED3	*+ED2	*+ED1
G120	*-ED8	*-ED7	*-ED6	*-ED5	*-ED4	*-ED3	*-ED2	*-ED1
G202	NDCAL8	NDCAL7	NDCAL6	NDCAL5	NDCAL4	NDCAL3	NDCAL2	NDCAL1
G341	*+ED48	*+ED47	*+ED46	*+ED45	*+ED44	*+ED43	*+ED42	*+ED41
G342	*-ED48	*-ED47	*-ED46	*-ED45	*-ED44	*-ED43	*-ED42	*-ED41
G343	*+ED58	*+ED57	*+ED56	*+ED55	*+ED54	*+ED53	*+ED52	*+ED51
G344	*-ED58	*-ED57	*-ED56	*-ED55	*-ED54	*-ED53	*-ED52	*-ED51
G352	*FHRO7	*FHRO6	*FHRO5	*FHRO4	*FHRO3	*FHRO2	*FHRO1	*FHRO0
G353	FHROV						*FHRO9	*FHRO8

CNC → PMC

The signals below are output signals related to PMC axis control.
For axis-type signals, addresses for the 1st axis to 8th axis are indicated.

	#7	#6	#5	#4	#3	#2	#1	#0
F094	ZP8	ZP7	ZP6	ZP5	ZP4	ZP3	ZP2	ZP1
F096	ZP28	ZP27	ZP26	ZP25	ZP24	ZP23	ZP22	ZP21
F098	ZP38	ZP37	ZP36	ZP35	ZP34	ZP33	ZP32	ZP31
F100	ZP48	ZP47	ZP46	ZP45	ZP44	ZP43	ZP42	ZP41

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0000						INI		

[Input type] Setting input
[Data type] Bit path

#2 INI Unit of input
0: In metrics
1: In inches

	#7	#6	#5	#4	#3	#2	#1	#0
0012								MIRx

[Input type] Setting input
[Data type] Bit axis

#0 MIRx Mirror image for each axis
0: Mirror image is off. (Normal)
1: Mirror image is on. (Mirror)

	#7	#6	#5	#4	#3	#2	#1	#0
1001								INM

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 INM Least command increment on the linear axis
0: In mm (metric system machine)
1: In inches (inch system machine)

	#7	#6	#5	#4	#3	#2	#1	#0
1005							DLZx	

[Input type] Parameter input
[Data type] Bit axis

#1 DLZx Function for setting the reference position without dogs
0: Disabled
1: Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMIx		DIAx			

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#3 DIAx The move command for each axis is based on:
0: Radius specification
1: Diameter specification

#5 ZMIx The direction of manual reference position return is:
0: + direction
1: - direction

	#7	#6	#5	#4	#3	#2	#1	#0
1008							RABx	ROAx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROAx The rotary axis roll-over is
0: Invalid
1: Valid

> **NOTE**
> ROAx specifies the function only for a rotary axis (for which bit 0 (ROTx) of parameter No. 1006 is set to 1)

#1 RABx In the absolute programming, the axis rotates in the direction
0: In which the distance to the target is shorter.
1: Specified by the sign of command value.

> **NOTE**
> RABx is valid only when ROAx is 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1201								ZPR

[Input type] Parameter input
[Data type] Bit path

#0 ZPR Automatic setting of a coordinate system when the manual reference position return is performed
0: Not set automatically
1: Set automatically

> **NOTE**
> ZPR is valid when the workpiece coordinate system is not used (when bit 0 (NWZ) of parameter No. 8136 is 1). If a workpiece coordinate system function is provided, making a manual reference position return always causes the workpiece coordinate system to be established on the basis of the workpiece zero point offset (parameters Nos. 1220 to 1226), irrespective of this parameter setting.

1240	Coordinate value of the reference position in the machine coordinate system

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the reference position in the machine coordinate system.

1241	Coordinate value of the second reference position in the machine coordinate system
1242	Coordinate value of the third reference position in the machine coordinate system
1243	Coordinate value of the fourth reference position in the machine coordinate system

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the second to fourth reference positions in the machine coordinate system.

1250	Coordinate system of the reference position used when automatic coordinate system setting is performed

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the coordinate system of the reference position on each axis to be used for performing automatic coordinate system setting.

1260	The shift amount per one rotation of a rotary axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

Set the shift amount per one rotation of a rotary axis.
For the rotary axis used for cylindrical interpolation, set the standard value.

	#7	#6	#5	#4	#3	#2	#1	#0
1402							JOV	

[Input type] Parameter input
[Data type] Bit path

#1 JOV Jog override is:
0: Enabled
1: Disabled (tied to 100%)

1420	Rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1421	F0 rate of rapid traverse override for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the F0 rate of the rapid traverse override for each axis.

1424	Manual rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rate of manual rapid traverse when the rapid traverse override is 100% for each axis.

NOTE
1 If 0 is set, the rate set in parameter No. 1420 (rapid traverse rate for each axis) is assumed.
2 When manual rapid traverse is selected (bit 0 (RPD) of parameter No. 1401 is set to 1), manual feed is performed at the feedrate set in this parameter, regardless of the setting of bit 4 (JRV) of parameter No. 1402.

1430	Maximum cutting feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Specify the maximum cutting feedrate for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1803				TQF				

[Input type] Parameter input
[Data type] Bit path

#4 TQF When torque control is performed by the PMC axis control, follow-up operation is:
0: Not performed.
1: Performed.

	#7	#6	#5	#4	#3	#2	#1	#0
1805							TRE	

[Input type] Parameter input
[Data type] Bit path

#1 TRE When bit 4 (TQF) of parameter No. 1803 is set to 0 (not to perform follow-up operation with a torque control command in PMC axis control), the servo error counter is:

0: Updated.
When the error count exceeds the maximum allowable cumulative travel value (parameter No. 1885), the alarm SV0423, “EXCESS ERROR INTORQUE” is issued.

1: Not updated.
No errors are accumulated, so that the alarm SV0423 is not issued. When the maximum allowable feedrate is exceeded, however, the alarm SV0422, “EXCESS VELOCITY IN TORQUE” is issued.

To return to position control when this parameter bit is set to 1, a reference position return operation needs to be performed.

1826	In-position width for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

The in-position width is set for each axis.
When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position. (The machine is in the in-position state.)

1827	In-position width in cutting feed for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set an in-position width for each axis in cutting feed. This parameter is used when bit 4 (CCI) of parameter No. 1801=1.

1836	Servo error amount where reference position return is possible

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] 0 to 32767

This parameter sets a servo error used to enable reference position return.
In general, set this parameter to 0. (When 0 is set, 128 is assumed as the default.)

When the servo error amount never exceeds this parameter value before leaving the limit switch for the deceleration in reference position return, alarm (PS0090), "REFERENCE RETURN INCOMPLETE" occurs.

1885	Maximum allowable value for total travel during torque control

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] 0 to 32767

Set a maximum allowable cumulative travel value (error counter value) during torque control. If the cumulative travel value exceeds the set value, the servo alarm SV0423, “EXCESS ERROR INTORQUE” is issued.

> **NOTE**
> This parameter is enabled when the bit 4 (TQF) of parameter No. 1803 is 0 (follow-up is not performed during torque control).

1886	Positional deviation when torque control is canceled

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] 1 to 32767

Set a positional deviation value when torque control is canceled to return to positional deviation. After the positional deviation has fallen to the parameter-set value, switching to position control is performed.

> **NOTE**
> This parameter is enabled when the bit 4 (TQF) of parameter No. 1803 is 0 (follow-up is not performed during torque control).

	#7	#6	#5	#4	#3	#2	#1	#0
2000							DGP	

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit axis

#1 DGP Upon power-up, the digital servo parameter specific to a motor is:
0: Set.
1: Not set.

When this parameter is set to 0 after the motor type is set, the digital servo parameter is automatically set to the standard value appropriate for the motor type and this parameter is set to 1 at the same time.

	#7	#6	#5	#4	#3	#2	#1	#0
2007	TRQ							

[Input type] Parameter input
[Data type] Bit axis

#7 TRQ Torque control is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
2017				HTN				

[Input type] Parameter input
[Data type] Bit axis

#4 HTN In the speed command mode, the hardware broken wire alarm for the separate detector is:
0: Detected.
1: Ignored.

2105	Torque constant

[Input type] Parameter input
[Data type] Bit axis
[Unit of data] 0.00001Nm/ (torque command)
[Valid data range] 1 to 32767
This parameter is set for each torque characteristic.

> **NOTE**
> When a linear motor is used, the data unit is 0.001 N/(1 torque command).

	#7	#6	#5	#4	#3	#2	#1	#0
3002				IOV				

[Input type] Parameter input
[Data type] Bit path

#4 IOV Override-related signal logic is:
0: Used without modification
(A signal of negative logic is used as a negative logic signal, and a signal of positive logic is used as a positive logic signal.)
1: Inverted
(A signal of negative logic is used as a positive logic signal, and a signal of positive logic is used as a negative logic signal.)

The signals indicated below are affected.
Signal of negative logic:
- Feedrate override signals *FV0 to *FV7<Gn012>
- Second feedrate override signals*AFV0 to *AFV7<Gn013>
- 0.01% step second feedrate override signals*APF00 to *APF15<Gn094, Gn095>
- Feedrate override signals (for PMC axis control)
  *EFOV0g to *EFOV7g<G0151/G0163/G0175/G0187>

- Software operator's panel signals *FV0O to *FV7O<Fn078>

Signals of positive logic:

- Rapid traverse override signals ROV1,ROV2<Gn014.0, Gn014.1>
- Software operator's panel signals ROV1O,ROV2O<Fn076.4, Fn076.5>
- Rapid traverse override signals (for PMC axis control) EROV1g,EROV2g<G0150.0, G0150.1/G0162.0, G0162.1/G0174.0, G0174.1/G0186.0, G0186.1>

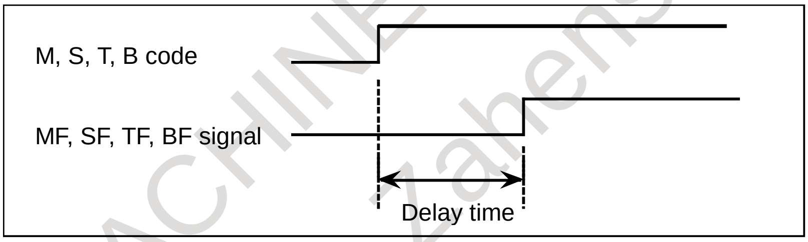
The signals indicated below are not affected.

- 1% step rapid traverse override selection signal HROV <Gn096.7>
- 1% step rapid traverse override signals *HROV0 to *HROV6 <Gn096.0 to Gn096.6>
- 0.1% step rapid traverse override selection signal FHROV <Gn353.7>
- 0.1% step rapid traverse override signals *FHRO0 to *FHRO9 <Gn352.0 to Gn352.7, Gn353.0 to Gn353.1>

3010	Time lag in strobe signals MF, SF, TF, and BF

[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 32767

The time required to send strobe signals MF<Fn007.0>, SF<Fn007.2>, TF<Fn007.3>, and BF<Fn007.7> after the M, S, T, and B codes are sent, respectively.



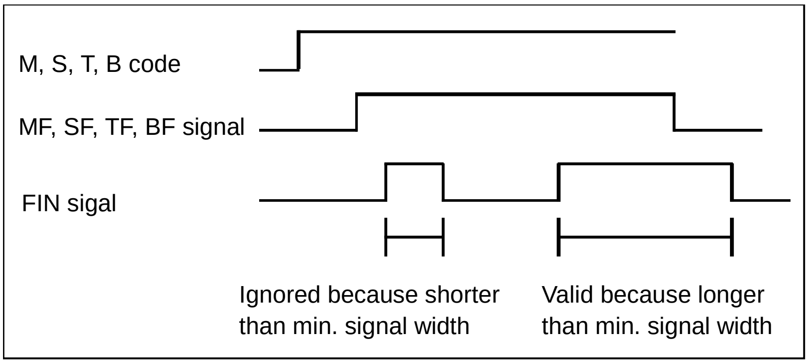
NOTE

The time is counted in units of 8 ms. If the set value is not a multiple of eight, it is raised to the next multiple of eight.
Example
When 30 is set, 32 ms is assumed.
When 0 is set, 8 ms is assumed.

3011	Acceptable width of M, S, T, and B function completion signal (FIN)

[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 32767

Set the minimum signal width of the valid M, S, T, and B function completion signal (FIN).



> **NOTE**
> The time is counted in units of 8 ms. If the set value is not a multiple of eight, it is raised to the next multiple of eight
> Example
> When 30 is set, 32ms is assumed.
> When 0 is set, 8ms is assumed.
> The time count period may change, depending on the system.

	#7	#6	#5	#4	#3	#2	#1	#0
3104					PPD			

[Input type] Parameter input
[Data type] Bit path

#3 PPD Relative position display when a coordinate system is set
0: Not preset
1: Preset

> **NOTE**
> If any of the following is executed when PPD is set to 1, the relative position display is preset to the same value as the absolute position display:
> (1) Manual reference position return
> (2) Coordinate system setting based on G92 (G50 for G code system A on the lathe system)
> (3) Workpiece coordinate system presetting based on G92.1 (G50.3 for G code system A on the lathe system)
> (4) When a T code for the lathe system is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
3105							PCF	

[Input type] Parameter input
[Data type] Bit path

#1 PCF Addition of the movement of the PMC-controlled axes to the actual speed display
0: Added
1: Not added

	#7	#6	#5	#4	#3	#2	#1	#0
3115					NDFx			

[Input type] Parameter input
[Data type] Bit axis

#3 NDFx In calculation for actual cutting feedrate display, the feedrate of a selected axis is:
0: Considered.
1: Not considered.

	#7	#6	#5	#4	#3	#2	#1	#0
8001	SKE	AUX	NCC		RDE	OVE		MLE

[Input type] Parameter input
[Data type] Bit path

#0 MLE Whether all axis machine lock signal MLK <Gn044.1> is valid for PMC-controlled axes
0: Valid
1: Invalid
The axis-by-axis machine lock signal MLKx <Gn108> depends on the setting of bit 1 of parameter No. 8006.

#2 OVE Signals related to dry run and override used in PMC axis control
0: Same signals as those used for the CNC
1: Signals specific to the PMC
The signals used depend on the settings of these parameter bits as indicated below.

Signals	Bit 2 (OVE) of parameter No. 8001 = 0 (same signals as those used for the CNC)		Bit 2 (OVE) of parameter No. 8001 = 1 (signals specific to the PMC)	
Feedrate override signals	*FV0 to *FV7	<G012>	*EFOV0 to *EFOV7	<G151>
Override cancellation signal	OVC	<G006.4>	EOVC	<G150.5>
Rapid traverse override signals	ROV1,2	<G014.0,1>	EROV1,2 or *EROV0 to *EROV7	<G150.0,1> <G151>
Dry run signal	DRN	<G46.7>	EDRN	<G150.7>
Manual rapid traverse selection signal	RT	<G19.7>	ERT	<G150.6>

(The listed signal addresses when PMC signals are selected are for the 1st group. Actual addresses differs depending on the used group.)

> **NOTE**
> When PMC control is performed with more than four times as many groups as the number of NC paths, the signals shown in "same signals as those used for the CNC" cannot be used for the excess paths and groups. Therefore, set this parameter to 1 for this case.

#3 RDE Whether dry run is valid for rapid traverse in PMC axis control
0: Invalid
1: Valid

#5 NCC When the program specifies a move command for a PMC-controlled axis (with the controlled axis selection signal EAX <G0136> set to “1”) not placed under PMC axis control:
0: CNC command is valid.
1: The alarm PS0130, “NC AND PMC AXIS ARE CONFLICTED” is issued.

#6 AUX In PMC axis control, the auxiliary function command (12H) output size is:
0: 1 byte (0 to 255)
1: 2 bytes (0 to 65535)

#7 SKE Skip signal during axis control by the PMC
0: Uses the same signal SKIP <X004.7, X013.7, or X011.7> as CNC.
1: Uses dedicated axis control signal ESKIP <X004.6, X013.6, or X011.6> used by the PMC.

> ⚠ **WARNING**
> If signal ESKIP<X004.6, X013.6, or X011.6> is used by another usage when you set 1 to this parameter, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used X address.

	#7	#6	#5	#4	#3	#2	#1	#0
8002	FR2	FR1	PF2	PF1	F10		DWE	RPD

[Input type] Parameter input
[Data type] Bit path

#0 RPD Rapid traverse rate for PMC-controlled axes
0: Feedrate specified with parameter No. 1420
1: Feedrate specified with the feedrate data in an axis control command by PMC

#1 DWE Minimum time which can be specified in a dwell command in PMC axis control when the increment system is IS-C
0: 1ms
1: 0.1ms

#3 F10 Least increment for the feedrate for cutting feed (per minute) in PMC axis control
The following settings are applied when bit 4 (PF1) of parameter No. 8002 is set to 0 and bit 5 (PF2) of parameter No. 8002 is set to 0.

	F10	IS-A	IS-B	IS-C
Millimeter input (mm/min)	0	10	1	0.1
	1	100	10	1
Inch input (inch/min)	0	0.1	0.01	0.001
	1	1	0.1	0.01

#4 PF1
#5 PF2 Set the feedrate unit of cutting feedrate (feed per minute) for an axis controlled by the PMC.

Bit 5 (PF2) of parameter No. 8002	Bit 4 (PF1) of parameter No. 8002	Feedrate unit
0	0	1 / 1
0	1	1 / 10
1	0	1 / 100
1	1	1 / 1000

#6 FR1
#7 FR2 Set the feedrate unit for cutting feedrate (feed per rotation) for an axis controlled by the PMC.

(1)T series
[Unit of data]
Unit of data depending on the setting of bits 6 (FR1) and bit 7 (FR2) of parameter No. 8002.

Parameter		Metric input (mm/rev)	Inch input (inch/rev)	Rotation axis (deg/rev)
FR2	FR1			
1	1	0.0001	0.000001	0.0001
0	0			
0	1	0.001	0.00001	0.001
1	0	0.01	0.0001	0.01

[Valid data range] 1 to 65535 (However, data within the ranges indicated following table must be specified.)

		Valid data range	Unit
		IS-A to IS-C	
Linear axis	Metric input	0.0001 to 500.0000	mm/rev
	Inch input	0.000001 to 9.999999	inch/rev
Rotation axis		0.0001 to 500.0000	deg/rev

(2)M series
[Unit of data]
Unit of data depending on the setting of bits 6 (FR1) and bit 7 (FR2) of parameter No. 8002.

Parameter		Metric input (mm/rev)	Inch input (inch/rev)	Rotation axis (deg/rev)
FR2	FR1			
1	1	0.01	0.0001	0.01
0	0			
0	1	0.1	0.001	0.1
1	0	1	0.01	1

[Valid data range] 1 to 65535 (However, data within the ranges indicated following table must be specified.)

		Valid data range	Unit
		IS-A to IS-C	
Linear axis	Metric input	0.01 to 500.00	mm/rev
	Inch input	0.0001 to 9.9999	inch/rev
Rotation axis		0.01 to 500.00	deg/rev

	#7	#6	#5	#4	#3	#2	#1	#0
8003					FEX			

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter bit is set, the power must be turned off before operation is continued.

#3 FEX The maximum feedrate that can be achieved by the machine during cutting feed or continuous feed in PMC axis control, or spindle control with servo motor is:
0: Not extended.
1: Extended.

Restrictions

- Parameters for setting the time constants for linear acceleration/deceleration after interpolation and bell-shaped acceleration/deceleration after interpolation
  When as the acceleration/deceleration type, linear acceleration/ deceleration after interpolation or bell-shaped acceleration/ deceleration after interpolation is used for each of rapid traverse, cutting feed, and manual feed, the maximum allowable time constant is a half of the maximum value that can be set conventionally.
  The time constant parameters used are as follows:

Parameter No.	Meaning
1620	Time constant (T) used for linear acceleration/deceleration in rapid traverse for each axis, or time constant (T1) used for bell-shaped acceleration/deceleration in rapid traverse for each axis
1621	Time constant (T2) used for bell-shaped acceleration/deceleration in rapid traverse for each axis
1622	Time constant for acceleration/deceleration in cutting feed for each axis
1624	Time constant for acceleration/deceleration in jog feed for each axis
1626	Time constant for acceleration/deceleration in threading cycles for each axis
1769	Time constant for acceleration/deceleration after cutting feed interpolation in the mode of look-ahead acceleration/deceleration before interpolation
5271 to 5274	Time constant for acceleration/deceleration in rigid tapping extraction (first to fourth gears)
5365 to 5368	Time constant for bell-shaped acceleration/deceleration in rigid tapping (first to fourth gears)

- The waveform display function of VCMD on the SERVO GUIDE or the SERVO GUIDE Mate.
  As the feedrate increases, more data is acquired for VCMD waveform display, which can prevent waveforms from being displayed correctly.

⚠ CAUTION

1. When this function is enabled, the feedrate is extended to the maximum value that can be specified for cutting feed or continuous feed in PMC axis control if CMR is 1. If CMR is greater than 1, the feedrate is limited to a value smaller than the maximum value that can be specified.
2. Note that the maximum motor speed may be exceeded depending on the feedrate specified.

	#7	#6	#5	#4	#3	#2	#1	#0
8004		NCI	DSL			JFM		

[Input type] Parameter input
[Data type] Bit path

#2 JFM This parameter sets the units used to specify feedrate data when continuous feed is specified in axis control by the PMC.

Increment system	Bit 2 (JFM) of No. 8004	Millimeter input (mm/min)	Inch input (inch/min)	Rotation axis (deg/min)
IS-B	0	1	0.01	1
	1	200	2.00	200
IS-C	0	0.1	0.001	0.1
	1	20	0.200	20

#5 DSL If the selection of an axis is changed when PMC axis selection is disabled:
0: An alarm PS0139, "CANNOT CHANGE PMC CONTROL AXIS" is issued.
1: The change is valid, and no alarm is issued for an unspecified group.

#6 NCI In axis control by the PMC, a position check at the time of deceleration is:
0: Performed.
1: Not performed.

	#7	#6	#5	#4	#3	#2	#1	#0
8005			IFV	EVP	DRR	R10	CDI	EDC

[Input type] Setting input
[Data type] Bit path

#0 EDC In axis control by the PMC, an external deceleration function is:
0: Disabled.
1: Enabled.

#1 CDI In axis control by the PMC, when diameter programming is specified for a PMC-controlled axis:
0: The amount of travel and feedrate are each specified with a radius.
1: The amount of travel is specified with a diameter while the feedrate is specified with a radius.
This parameter is valid when bit 3 (DIA) of parameter No. 1006 is set to 1 (A move command for each axis is based on diameter specification.)

#2 R10 When the bit 0 (RPD) of parameter No. 8002 is set to 1, the unit for specifying a rapid traverse rate for the PMC axis is:
0: 1 mm/min.
1: 10mm/min.

#3 DRR For cutting feed per rotation in PMC axis control, the dry run function is:
0: Disabled.
1: Enabled.

#4 EVP Speed command in PMC axis control is executed by:
0: Velocity control.
1: Position control.
This bit is available when speed command in PMC axis control is FS16 type (bit 2 (VCP) of parameter No. 8007 is 1).

#5 IFV When bit 2 (OVE) of parameter No. 8001 is set to 1 in PMC axis control, the feedrate override signal *EFOVx and the override cancel signal OVC are:
0: Used on a path-by-path basis. (The start groups (1st group, 5th group, 9th group, 13th group) of each path are used.)
1: Used on a group-by-group basis.

	#7	#6	#5	#4	#3	#2	#1	#0
8006		EZR		EFD			MLS	

[Input type] Parameter input
[Data type] Bit path

#1 MLS When bit 0 (MLE) of parameter No. 8001 is set to 1 (to disable the all axis machine lock signal) in PMC axis control, axis-by-axis machine lock is:
0: Disabled.
1: Enabled.

#4 EFD When cutting feed (feed per minute) is used in PMC axis control, the specification unit of feedrate data is:
0: Unchanged (1 times).
1: 100 times greater.

> **NOTE**
> When this parameter is set to 1, bit 3 (F10) of parameter No. 8002 is invalid.

#6 EZR In PMC axis control, bit 0 (ZRNx) of parameter No. 1005 is:
0: Invalid.
With a PMC controlled axis, the alarm PS0224, "ZERO RETURN NOT FINISHED" is not issued.
1: Valid.
A reference position return state check is made on a PMC controlled axis as with an NC axis according to the setting of bit 0 (ZRNx) of parameter No. 1005.

	#7	#6	#5	#4	#3	#2	#1	#0
8007			PSA		ESY	VCP		

[Input type] Parameter input
[Data type] Bit path

#2 VCP Speed command in PMC axis control is:
0: FS15 type.
1: FS16 type.

#3 ESY In PMC axis control, external pulse synchronization (serial spindle synchronization) is:
0: Disabled.
1: Enabled.

#5 PSA Acceleration/deceleration after interpolation for skip command of PMC axis control is
0: Invalid
1: Valid

> **NOTE**
> Skip position of system variables (#5061 to #5080, #100151 to #100182) are behavior such as the following.
> - In case of parameter PSA = 0, coordinate value of bit 1 (SEB) of parameter No.6201= 0 equivalent is stored.
> - In case of parameter PSA = 1, coordinate value in accordance with the setting of parameter SEB is stored.

	#7	#6	#5	#4	#3	#2	#1	#0
8008								**EMRx**

[Input type] Parameter input
[Data type] Bit axis

#0 EMRx When a PMC axis control command is issued in mirror image state, the mirror image is:
0: Not considered.
1: Considered.

This parameter is valid in the mirror image mode set with the mirror image signals MI1 to MI8 <G106.0 to .7> set to 1 or bit 0 (MIRx) of parameter No. 12 set to 1.
If a movement is made along the same axis by doubly specifying a command with the CNC and PMC axis control when this parameter is set to 0, and the mirror image mode is set, a coordinate shift can occur afterwards. So, do not attempt to make such a movement.

8010	Selection of the DI/DO group for each axis controlled by the PMC

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 1 to 16

Specify the DI/DO group to be used to specify a command for each PMC-controlled axis.

For addresses of the fifth group and up, 1000 is added in steps of 4 groups.
For example:
The start address of the 10th group is G2154.

Parameter No.8010	Description
1	DI/DO 1st group <G142 to G153> is used.
2	DI/DO 2nd group <G154 to G165> is used.
3	DI/DO 3rd group <G166 to G177> is used.
4	DI/DO 4th group <G178 to G189> is used.
5	DI/DO 5th group <G1142 to G1153> is used.
6	DI/DO 6th group <G1154 to G1165> is used.
:	:
13	DI/DO 13th group <G3142 to G3153> is used.
14	DI/DO 14th group <G3154 to G3165> is used.
15	DI/DO 15th group <G3166 to G3177> is used.
16	DI/DO 16th group <G3178 to G3189> is used.

NOTE
When a value other than the above is set, the axis is not controlled by the PMC.

	#7	#6	#5	#4	#3	#2	#1	#0
8013				R20x	ROP		OVR	

[Input type] Parameter input
[Data type] Bit axis

#1 OVR When bit 2 (OVE) of parameter No. 8001 is set to 1, for rapid traverse command override in PMC axis control:

0: Rapid traverse override signals EROV2 and EROV1 <G150.1 and G150.0> for PMC axis control are used.
1: 1% step rapid traverse override signals *EROV7 to *EROV0 <G151> for PMC axis control are used.

(The listed signal addresses when PMC signals are selected are for the 1st group. Actual addresses differs depending on the used group.)

#3 ROP When rotation axis rollover is enabled for an axis controlled in PMC axis control, the direction in which a movement (rotation) is performed to reach an end point by a reference position return command 07H to 0AH (equivalent to G28, G30P2/P3/P4) is:

0: Determined by the sign of the commanded value.
1: The direction in the shortest path.

> **NOTE**
> ROPx is valid only when bit 0 (ROAx) of parameter No. 1008 is set to 1 and bit 1 (RABx) of parameter No. 1008 is set to 0.

#4 R20x When the machine coordinate system selection (20h) is commanded with the PMC axis control for the rotary axis to which the roll-over function is valid (bit 0 (ROAx) of parameter No. 1008 is set to 1), setting the bit 1 (RABx) of parameter No. 1008 which specifies the direction of the rotation for an absolute command is:

0: Invalid.
1: Valid.

The above direction of the rotation is as follows by setting the bit 1 (RABx) of parameter No. 1008 and the bit 4 (R20x) of parameter No. 8013.

<table>
<tr><th colspan="2" rowspan="2"></th><th colspan="2">Bit 4 (R20x) of parameter No.8013</th></tr>
<tr><th>0</th><th>1</th></tr>
<tr><td rowspan="2">Bit 1 (RABx) of parameter No.1008</td><td>0</td><td>Direction of the shortest path</td><td>Direction of the shortest path</td></tr>
<tr><td>1</td><td>Direction of sign of the amount of the movement to be made</td><td>Direction of sign of the command value</td></tr>
</table>

	#7	#6	#5	#4	#3	#2	#1	#0
8019						EZC	PIA	EOS

[Input type] Parameter input
[Data type] Bit

#0 EOS In external pulse synchronization (serial spindle synchronization) in PMC axis control, the serial spindle to be synchronized is:

0: The first spindle of path 1.
1: Any spindle.

> **NOTE**
> If Bit 0 (EOS) of parameter No. 8019 is set to 0, only the servo axis of path 1 can be commanded.

#1 PIA After the move command of PMC axis control is interrupted by reset signal ECLRg, if different acceleration/deceleration type command without waiting for a deceleration stop generates:
0: Not alarm.
1: Alarm DS1451, "IMPROPER PMC AXIS COMMAND".

> **NOTE**
> This parameter is automatic setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

#2 EZC If the controlled axis selection signals EAX1 to EAX8 <Gn136> are "0" or PMC controlled-axis selection variable (#8700) is 0, the PMC axis control command generates:
0: Not alarm.
1: Alarm DS1451, "IMPROPER PMC AXIS COMMAND".

> **NOTE**
> This parameter is automatic setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

8020	FL rate for reference position return along each axis in PMC axis control

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
For each axis, this parameter sets a feedrate (FL rate) after deceleration for reference position return in PMC axis control.

> **NOTE**
> If 0 is specified, the value of parameter No. 1425 is used.

8022	Upper limit rate of feed per revolution during PMC axis control

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the upper limit rate of feed per revolution during PMC axis control.

8028	Time for acceleration/deceleration calculation when a feedrate is commanded under PMC axis control

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 32767

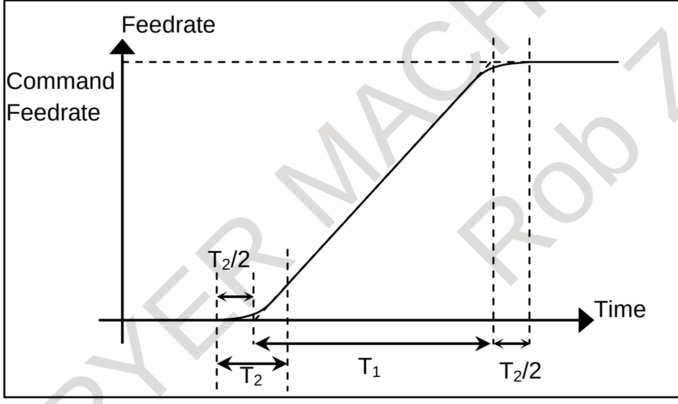
When a feedrate is commanded under PMC axis control, acceleration/deceleration can be set for parameter No. 8032 or this parameter. When 0 is set in parameter No. 8032, the specification of 1000 min^{-1} is assumed. When 0 is set in this parameter, the acceleration/deceleration function for feedrate specification is disabled.

8029	Acceleration change time of bell-shaped acceleration/deceleration for the speed command under PMC axis control

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 512

Set an acceleration change time of bell-shaped acceleration/deceleration for the speed command under PMC axis control for each axis.
To enable bell-shaped acceleration/deceleration, acceleration/deceleration function for the speed command must be valid (Parameter (No. 8028) is set to other than 0). When the acceleration/deceleration is enabled, set the acceleration change time corresponding to the acceleration in this parameter.

If this parameter is set to 0 or less of the value, bell-shaped acceleration/deceleration is disabled.
If this parameter is set 512 or more, the value is clamped to 512.



T_1 : Acceleration time in the acceleration that has been set by linear acceleration/deceleration
(This time will vary depending on the setting and command value of speed command)
T_2 : Acceleration change time (Parameter No. 8029)
Total time = $T_1 + T_2$
Time for the linear section = $T_1 - T_2$
Time for the bell-shaped section = T_2

8030	Time constant for exponential acceleration/deceleration in cutting feed or continuous feed under PMC axis control

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] msec
[Valid data range] 0 to 4000

For each axis, this parameter sets a time constant for exponential acceleration/deceleration in cutting feed or continuous feed under PMC axis control.

> **NOTE**
> When either of the following conditions is satisfied, the value set in parameter No. 1622 is used.
> - 0 is set in this parameter.
> - Acceleration/deceleration after cutting interpolation is linear type.
> - Advanced preview feed forward is enabled.

8031	FL rate for exponential acceleration/deceleration in cutting feed or continuous feed under PMC axis control

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

For each axis, this parameters sets a lower feedrate limit (FL rate) for exponential acceleration/deceleration in cutting feed or continuous feed under PMC axis control.

> **NOTE**
> 1 When either of the following conditions is set, the value set in parameter No. 1623 is used.
> - 0 is set in this parameter.
> - Advanced preview feed forward is enabled.
>
> 2 Be sure to set 0 in this parameter and parameter No. 1623 for all axes at all times except for special purposes. If a value other than 0 is specified, correct linear or circular figures cannot be obtained.

8032	Feedrate for acceleration/deceleration calculation when a feedrate is commanded under PMC axis control

[Input type] Parameter input
[Data type] Word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 32767

When a feedrate is specified under PMC axis control, acceleration/deceleration can be set for this parameter or parameter No. 8028. When 0 is set in this parameter, the specification of 1000 min^{-1} is assumed. When 0 is set in parameter No. 8028, the acceleration/deceleration function for feedrate specification is disabled.

8040	Amount of a shift per one rotation of a servo motor of least input increment when speed command in PMC axis control is position control

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] mm, inch, deg (machine unit)
[Valid data range] 1 to 99999999

Set the amount of a shift per one rotation of a servo motor of least input increment when speed command in PMC axis control is position control.
This parameter is available when speed command in PMC axis control is FS16 type (bit 2 (VCP) of parameter No. 8007 is 1) and is executed by position control (bit 4 (EVP) of parameter No. 8005 is 1).

	#7	#6	#5	#4	#3	#2	#1	#0
11216						TPM		

[Input type] Parameter input
[Data type] Bit path

#2 TPM In PMC axis control and 3-dimensional coordinate system conversion / Tilted working plane indexing, NC command not involving a movement :

0: Disable to command (Alarm (PS5131) "NC COMMAND IS NOT COMPATIBLE" occurs).
1: Enabled to command.

> **NOTE**
> 1 When this parameter bit is set, the power must be turned off before operation is continued.
> 2 When the coordinate conversion axis (X,Y,Z) is PMC axis control mode, this function is disabled in following program command block.
> - Tilted working plane indexing start block (G68.2)
> - 3-dimensional coordinate start block (G68)
> - Tilted working plane indexing / 3-dimensional coordinate cancel block (G69)

	#7	#6	#5	#4	#3	#2	#1	#0
11850	IFH							CMI

[Input type] Parameter input
[Data type] Bit path

#0 CMI If, in PMC axis control, a rapid traverse rate is commanded with the axis control block data signal, with bit 0 (RPD) of parameter No. 8002 being set to 1, the rapid traverse rate is:

0: Always treated as being in millimeters.
1: Dependent on the setting of bit 0 (INM) of parameter No. 1001.

#7 IFH When bit 2 (OVE) of parameter No. 8001 is set to 1 in PMC axis control, the 1% rapid traverse override signals *EROVs are:

0: On a path-by-path basis. (The first groups in the individual paths (1st, 4th, 9th, 13th groups) are used.)
1: On a group-by-group basis.

Depending on this parameter and bit 1 (OVR) of parameter No. 8013, selected signals are as given in the table below.
(The signal addresses are those in the first path, and the actual addresses differ depending on the group used.)

	Bit 7 (IFH) of No. 11850 = 0 (*EROVs are on a path-by-path basis.)	Bit 7 (IFH) of No. 11850 = 1 (*EROVs are on a group-by-group basis.)
Bit 1 (OVR) of No. 8013 = 0	EROV1, EROV2 <G150.0, G150.1>	EROV1, EROV2 <G150.0, G150.1>
Bit 1 (OVR) of No. 8013 = 1	*EROV<G151>	*EROVA<G151> *EROVB<G163> *EROVC<G175> *EROVD<G187>

NOTE
Overrides are clamped at up to 100%.

	#7	#6	#5	#4	#3	#2	#1	#0
12730								**PTC**

[Input type] Parameter input
[Data type] Bit path

#0 PTC Linear acceleration/deceleration time constant of continuous feed operation based on a speed command in PMC axis control is:
0: Normal.
1: Extended.
This bit is available when speed command in PMC axis control is FS16 type (bit 2 (VCP) of parameter No. 8007 is 1).

12731	**2nd time constant of linear acceleration/deceleration of continuous feed operation based on a speed command in PMC axis control**

12732	**3rd time constant of linear acceleration/deceleration of continuous feed operation based on a speed command in PMC axis control**

12733	**4th time constant of linear acceleration/deceleration of continuous feed operation based on a speed command in PMC axis control**

12734	**5th time constant of linear acceleration/deceleration of continuous feed operation based on a speed command in PMC axis control**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec/1000min^{-1}
[Valid data range] 0 to 32767

If 0 is specified, the time constant at a given feedrate becomes invalid, and acceleration/deceleration is not performed.
This parameter is available when speed command in PMC axis control is FS16 type (bit 2 (VCP) of parameter No. 8007 is 1) and linear acceleration/deceleration time constant of continuous feed operation based on a speed command in PMC axis control is extended (bit 0 (PTC) of parameter No. 12730 is 1).

12735	**1st feedrate for changing time constant of continuous feed operation based on a speed command in PMC axis control**

12736	**2nd feedrate for changing time constant of continuous feed operation based on a speed command in PMC axis control**

12737	**3rd feedrate for changing time constant of continuous feed operation based on a speed command in PMC axis control**

12738	**4th feedrate for changing time constant of continuous feed operation based on a speed command in PMC axis control**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 32767

Set feedrate parameters as following.
No. 12735 < No. 12736 < No. 12737 < No. 12738.
This parameter is available when speed command in PMC axis control is FS16 type (bit 2 (VCP) of parameter No. 8007 is 1) and linear acceleration/deceleration time constant of continuous feed operation based on a speed command in PMC axis control is extended (bit 0 (PTC) of parameter No. 12730 is 1).

	#7	#6	#5	#4	#3	#2	#1	#0
18050	**OTW**							

[Input type] Parameter input
[Data type] Bit path

#7 OTW If an axis move command is executed with PMC axis control during automatic operation, and the NC block under execution is stopped by a feed hold when the axis moving due to PMC axis control is completed, the amount of movement due to PMC axis control in that block is:
0: Not reflected in the NC coordinate system.
1: Reflected in the NC coordinate system.

Alarm and message

Servo and overtravel alarms for PMC axis control are detected in the same way as for CNC controlled axes.
When an alarm is issued, the alarm signal EIALg is set to “1” for notification to the PMC in addition to normal alarm processing.
(When an overtravel alarm is issued, the negative-direction overtravel signal EOTNg or the positive-direction overtravel signal EOTPg is also set to “1”.)

The alarm signal EIALg is reset using the reset signal ECLRg.

> ⚠ **CAUTION**
> If an alarm is issued by a PMC axis control command or in connection with PMC axis control, be sure to set the reset signal ECLRg to “1” in addition to a reset operation on the NC.

- Alarms related to conflict with the NC

Alarm PS0130

In general, if the CNC and PMC attempt to simultaneously control an axis not subject to superimposition, the alarm PS0130 is issued.

This alarm is issued in the following cases:

- When cutting feed with an override of 0% is performed along the axis, or the tool is temporarily stopped along the axis (with the axis control temporary stop signal ESTPg set to "1"), this alarm is issued. (During feed hold or single block stop, a PMC axis control command is valid, not resulting in an alarm.)
- When a command is issued from the CNC for an axis controlled by the PMC
- When a move command is issued from the PMC for an axis on the polar coordinate interpolation plane in the polar coordinate interpolation mode (G12.1 command)

Alarm PS5131

If a command related to 3-dimensional coordinate conversion / tilted working plane indexing or polar coordinate interpolation is specified during PMC axis control, the alarm PS5131 is issued.

However, when 3-dimensional coordinate conversion / tilted working plane indexing mode and the bit 2 (TPM) of parameter No. 11216 is "1", CNC axis move commands to the same coordinate (no move command) can be commanded.

- Alarms due to setting modifications made during PMC axis control

Alarm PS0139

If parameter No. 8010 is modified during PMC axis control, or the level of the control axis selection signal EAXx for an axis under PMC axis control is changed, the alarm PS0139 is issued.

- Other alarms

Alarm PS0224

If a PMC axis control command is specified without performing a reference position return operation even once after the power is turned off when bit 0 (ZRNx) of parameter No. 1005 is set to 0, the alarm PS0224 is issued.

Alarm PS5065

If a PMC axis control command is executed by assigning axes with different increment systems to the same group, the alarm PS5065 is issued.

Number	Message	Description
PS0130	NC AND PMC AXIS ARE CONFLICTED	The NC command and the PMC axis control command were conflicted. Modify the program or ladder.
PS0139	CANNOT CHANGE PMC CONTROL AXIS	The PMC axis was selected for the axis for which the PMC axis is being controlled.
PS0224	ZERO RETURN NOT FINISHED	Reference position return has not been performed before the automatic operation starts. Perform reference position return only when the bit 0 (ZRNx) of parameter No. 1005 is set to 0.
PS5065	DIFFERRENT AXIS UNIT(PMC AXIS)	Axes having different increment systems have been set in the same DI/DO group for PMC axis control. Modify the setting of parameter No. 8010.
PS5131	NC COMMAND IS NOT COMPATIBLE	The PMC axis control and 3-dimensional coordinate conversion or a polar coordinate interpolation were specified simultaneously. Check the program or the ladder.
PS0220	ILLEGAL COMMAND IN SYNCHR-MODE	In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis. Modify the program or check the PMC ladder.
DS1184	PARAMETER ERROR IN TORQUE	An invalid parameter was set for torque control. The torque constant parameter is set to 0.

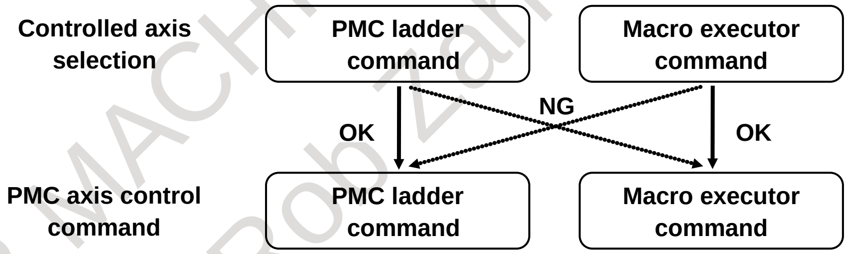
Number	Message	Description
SV0422	EXCESS VELOCITY IN TORQUE	In torque control, the commanded permissible velocity was exceeded.
SV0423	EXCESS ERROR IN TORQUE	In torque control, the maximum allowable cumulative travel value by the parameter was exceeded.
DS1451	IMPROPER PMC AXIS COMMAND	1. During deceleration of PMC axis control, next PMC axis control command is commanded. Correct PMC ladder sequence to execute the next command after the PMC axis control is stopped. 2. While the controlled axis selection signals EAX1 to EAX8 <Gn136> or PMC controlled-axis selection variable (#8700) are “0”, PMC axis control command is commanded. Correct PMC ladder sequence or macro executor program so that the signal or variable is set to 1.

Warning

⚠ WARNING

1 PMC axis control must be executed while the controlled axis selection signals EAX1 to EAX8 are "1". If controlled axis selection signals EAX1 to EAX8 are "0", the command cannot be accepted.

2 PMC axis control is able to command by PMC ladder and macro executor.
In case of use PMC axis control, “controlled axis selection” and “PMC axis control command”, use the same command method. If you use the different command method to “controlled axis selection” and “PMC axis control command”, “PMC axis control command” may be ignored, or the command may execute incorrect axis motion. Therefore, the machine may behave in an unexpected manner.



Caution

⚠ **CAUTION**

1 The mode selection, CNC reset, and other CNC statuses have no effect.
2 CNC-controlled feed hold *SP, single block stop SBK, reset signal ERS, and interlock of all axes or each axis are invalid for PMC axis control, but similar control is enabled by operating the PMC signals (ESTPg, ESBKg, and ECLRg).
3 Emergency stop is enabled.
4 If cutting feed blocks are specified in succession, deceleration is not performed between blocks (acceleration/deceleration is applied, however, when the specification of a feedrate changes), and cutting proceeds to the next block without waiting for the tool to enter the in-position area. At the end of a block other than for cutting feed, the tool is temporarily decelerated. The next block is then executed after waiting for the tool to enter the in-position area. When bit 6 (NCI) of parameter No. 8004 is set to 1, cutting can proceed to the next block without making an in-position check for each block.
5 Under PMC axis control, manual absolute mode is always set. If the PMC starts control of an axis after manual intervention (manual continuous feed, manual handle feed, etc.) is performed during automatic operation while manual absolute mode is not set (*ABSM is set to "1"), manual absolute mode is set.
6 Under PMC axis control, all commands are handled as axis commands. Even for the auxiliary functions, the position check is effective.
7 When the CNC executes the command to set the workpiece coordinate system setting (G54 to G59) during an axial movement by PMC axis control command, a valid coordinate system cannot be set.
8 If the alarm PS0139 is issued due to a modification to parameter No. 8010, the alarm is issued with all paths. After resetting the alarm, input the reset signal ECLRg for all groups before a start.

⚠ **CAUTION**

9 After the execution of a move command in PMC axis control during NC automatic operation, to execute a move command (in the absolute or incremental mode) for the relevant axis in an NC block during the same operation, execute the following method (1) or (2).
Note that in case of speed command or torque control, execute the method (2) because the method (1) can not be executed.

(1) Execute the PMC axis command by M code preventing buffering.

Execute the PMC axis command while the system is waiting for the FIN signal for an M code preventing buffering. Then, after the completion of that PMC axis command, return the FIN signal for the M code preventing buffering.
For example, in the following program, to execute the move command (N40) after the tool is moved along the Y-axis using a PMC axis command, the PMC axis command must be controlled within the M code preventing buffering (N20):
When the speed command (10h) or torque control command (11h) in PMC axis control is specified, execute the method (2) instead of this method.

O0001
N10 G94 G90 G01 X20. Y30. F3000 ;
N20 M55 ; → Y axis moves by PMC axis command.
N30 X70.0 ;
N40 Y50.0 ;
N50 M02 ;
Control under the following setting.

- Set an M code preventing buffering for the M55 block (Example: set 55 in parameter No. 3411).
- During M55 block, start PMC axis command. And, after the PMC axis command ends, the FIN signal for M55 is returned.

To perform the feed hold stop during PMC axis control, set bit 7 (OTW) of parameter No. 18050 to 1.

⚠ **CAUTION**

(2)-1 After the PMC axis command ends, command M code which turn on each axis workpiece coordinate system preset signal.
After the PMC axis command ends, command M code which turn on each axis workpiece coordinate system preset signal before the move command of NC block. Refer to "Each axis workpiece coordinate system preset signal" for details.

(2)-2 After the PMC axis command ends, command workpiece coordinate system preset.

As well as (2)-1, after the PMC axis command ends, command workpiece coordinate system preset before the move command of NC block.

For example, in the following program, to execute the move command(N50) after the tool is moved along the Y-axis using a PMC axis command, M code which turn on each axis workpiece coordinate system preset signal or workpiece coordinate system preset must be commanded in N40.

```
O0002
N10 G94 G90 G01 X20.0 Y30.0 F3000 ;
N20 M55 ; → Y axis moves start by PMC axis command.
N30 X70.0 ;
N40 M60 ; → M code which turn on each axis workpiece coordinate system
             reset signal is commanded.
            Or, workpiece coordinate system preset (G92.1Y0)
            is commanded.
N50 Y50. ;
N60 M02 ;
```

When the speed command (10h) or torque control command (11h) in PMC axis control is specified, this method is only applicable to the setting that the coordinates are updated.

10 Both feed-forward and advanced preview feed-forward function are unavailable for PMC axis.

11 For the axis under Cs contour control, PMC axis control can be used only following command.

- Rapid traverse (00h)
- Cutting feed (01h)
- Continuous feed (06h)
- Machine coordinate system selection (20h)
- Speed command (10h) for FS16*i* type position control
  (bit 2 (VCP) of parameter No. 8007 is set to 1 and bit 4 (EVP) of parameter No. 8005 is set to 1)

Note

NOTE
1 The actual speed excluding the effect of movement along a PMC-controlled axis can be displayed if the bit 3 (ZDF) of parameter No. 3115 is set to 1.
2 If an absolute pulse coder is used, a specified reference position is retained in memory, even after the power is turned off.
3 For an index table indexing axis, no PMC axis control command can be specified.
4 The individual output of the auxiliary function is provided by adding a signal for individual output. The timing diagram of controlling and commanding the auxiliary function is not changed. The normal specifications of the auxiliary functions for PMC axis control function are applied.

16.1.2 Advanced preview feed forward for PMC axis control

Overview

Advanced preview feed forward can be enabled for the commands executed by PMC axis control.
As a result, servo delay that becomes large as the feedrate becomes fast is reduced, and the command follow-up performance is improved.

Explanation

By setting bit 3 (G8C) and bit 4 (G8R) of parameter No.8004, advanced preview feed forward can be enabled for the commands executed by PMC axis control. (Table 16.1.2 (a))

Table 16.1.2 (a) Parameter setting and advanced preview feed forward

Parameter		Advanced preview feed forward
Bit 3 (G8C) of No.8004	Bit 4 (G8R) of No.8004	
0	0	Disable
0	1	Disable
1	0	Enable for cutting feed (Disable for rapid traverse)
1	1	Enable for cutting feed and rapid traverse

It is necessary to set bit 1 (FEEDx) of parameter No.2005 and parameter No.2092.

Command list of PMC axis control

PMC axis control enables the commands indicated in Table "Command list" to be specified.

In Table "Command list", "Command" represents the axis control command signals (EC0g to EC6g).
For the detail of the axis control command signals (EC0g to EC6g), refer to "PMC CONTROL FUNCTION".

"Advanced preview feed forward" shows enable or disable of advanced preview feed forward.
- Disable
  Advanced preview feed forward is disabled.
- Enable (Cutting feed)
  Advanced preview feed forward is enabled when the parameters are set to enable for cutting feed.
- Enable (Rapid traverse)
  Advanced preview feed forward is enabled when the parameters are set to enable for rapid traverse.

Table 16.1.2 (b) Command list

No.	Command	Operation	Advanced preview feed forward
(1)	00h	Rapid traverse	Enable (Rapid traverse)
(2)	01h	Cutting feed - feed per minute	Enable (Cutting feed)
(3)	02h	Cutting feed - feed per revolution	Enable (Cutting feed)
(4)	03h	Skip - feed per minute	Disable
(5)	04h	Dwell	Disable
(6)	05h	Reference position return	Disable
(7)	06h	Continuous feed	Enable (Cutting feed)
(8)	07h	First reference position return	Enable (Rapid traverse) (NOTE 1)
(9)	08h	Second reference position return	Enable (Rapid traverse)
(10)	09h	Third reference position return	Enable (Rapid traverse)
(11)	0Ah	Fourth reference position return	Enable (Rapid traverse)
(12)	0Bh	External pulse synchronization - position coder	Enable (Cutting feed)
(13)	0Dh	External pulse synchronization - first manual handle	Enable (Cutting feed)
(14)	0Eh	External pulse synchronization - second manual handle	Enable (Cutting feed)
(15)	0Fh	External pulse synchronization - third manual handle	Enable (Cutting feed)
(16)	10h	Speed command	Disable (NOTE 2)
(17)	11h	Torque control	Disable
(18)	12h	Auxiliary function 1	Disable
(19)	14h	Auxiliary function 2	Disable
(20)	15h	Auxiliary function 3	Disable
(21)	20h	Machine coordinate system selection	Enable (Rapid traverse)
(22)	21h	Cutting feed - sec/block specification	Enable (Cutting feed)

NOTE

1 If the axis that satisfies the both following conditions exists, advanced preview feed forward is disabled for all axes that belong to the same group.
   - Bit 1 (DLZx) of parameter No. 1005 for the function for setting the reference position without dogs is set to 0.
   - Reference position return operation is not performed even once after the power is turned on.

2 If the both following conditions are satisfied, advanced preview feed forward is “Enabled (Cutting feed)”.
   - FS16 type (Bit 2 (VCP) of parameter No.8007 is set to 1)
   - Position control (Bit 4 (EVP) of parameter No.8005 is set to 1)

Limitation

- Composite control
  PMC axis control enabled advanced preview feed forward cannot be used for the axes under composite control.
  Alarm PS0563 "CANNOT USE PMC AXIS (ADFF)" occurs if PMC axis control enabled advanced preview feed forward is specified for the axes under composite control.

- Superimposed control
  When PMC axis control enabled advanced preview feed forward is used for the axes under superimposed control, it is necessary to set superimposed ahead signal OVLN <Gn531.4> to 1 in all paths related to superimposed control and place the paths in the superimposed ahead mode.
  Alarm PS0563 "CANNOT USE PMC AXIS (ADFF)" occurs if PMC axis control enabled advanced preview feed forward is specified for the axes under superimposed control not in the superimposed ahead mode.

NOTE

NOTE

1. In this function, the settings of bit 2 (AOFF) of parameter No.1611 and bit 3 (FFR) of parameter No.1800 are disabled.
2. When the same operation is performed on the multiple axes by assigning the axes to one group, set the parameters to be the same condition about advanced preview feed forward in the axes. If the settings are different in the axes, it may not be the same movement.
3. When advanced preview feed forward is enabled, the settings of parameter No.8030 and No.8031 are disabled.
4. When the cutting/rapid feed-forward switching function is used for PMC axis control, set bit 6 (NCI) of parameter No.8004 to 0, and check in-position in the appropriate in-position range.
5. By setting bit 1 (PFE) of parameter No.8008 to 1, it is also possible to enable advanced preview feed forward for PMC axis control. However, it is necessary to set superimposed ahead signal OVLN <Gn531.4> to "1" and set to the superimposition preceding mode. Also, the valid commands for advanced preview feed forward are limited to rapid traverse (00h), cutting feed - feed per minute (01h), cutting feed - feed per revolution (02h), and cutting feed - sec / block specification (21h). Since it is a parameter of the old specification, basically use bit3 (G8C) and bit 4 (G8R) of parameter No.8004.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8004		NCI		G8R	G8C			

[Input type] Parameter input
[Data type] Bit path

#3 G8C In PMC axis control, advanced preview feed forward is :
0: Disabled.
1: Enabled.

#4 G8R In PMC axis control, advanced preview feed forward is :
0: Enabled for cutting feed. (Disabled for rapid traverse.)
1: Enabled for cutting feed and rapid traverse.

#6 NCI In axis control by the PMC, a position check at the time of deceleration is:

0: Performed.
1: Not performed.

	#7	#6	#5	#4	#3	#2	#1	#0
8008							PFE	

[Input type] Parameter input
[Data type] Bit axis

#1 PEF If superimposed ahead signal OVLN <Gn531.4> is set to "1", advanced preview feed forward for PMC axis control rapid traverse (00h), cutting feed – feed per minute (01h), cutting feed – feed per revolution (02h), and cutting feed – sec/block specification (21h) is:

0: Disabled.
1: Enabled.

> **NOTE**
> 1 This parameter is valid for rapid traverse (00h) when bit3 (FFR) of parameter No. 1800 is 1 (advanced preview feed forward is enabled for rapid traverse).
> 2 This parameter is a parameter of the old specification. It is recommended not to use this parameter, but to use bit3 (G8C) and bit 4 (G8R) of parameter No.8004. If both this parameter and G8C are set to 1, G8C has priority.

8030	Time constant for exponential acceleration/deceleration in cutting feed or continuous feed under PMC axis control

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] msec
[Valid data range] 0 to 4000

For each axis, this parameter sets a time constant for exponential acceleration/deceleration in cutting feed or continuous feed under PMC axis control.

> **NOTE**
> When either of the following conditions is satisfied, this parameter is invalid and the value set in parameter No.1622 is used.
> - 0 is set in this parameter.
> - Acceleration/deceleration after cutting interpolation is linear type.
> - Advanced preview feed forward is enabled.

8031	FL rate for exponential acceleration/deceleration in cutting feed or continuous feed under PMC axis control

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

For each axis, this parameter sets a lower feedrate limit (FL rate) for exponential acceleration/deceleration in cutting feed or continuous feed under PMC axis control.

NOTE
1 When either of the following conditions is satisfied, this parameter is invalid and the value set in parameter No.1623 is used.
  - 0 is set in this parameter.
  - Advanced preview feed forward is enabled.
2 Be sure to set 0 in this parameter and parameter No. 1623 for all axes at all times except for special purposes. If a value other than 0 is set, correct linear or circular figures cannot be obtained.

	#7	#6	#5	#4	#3	#2	#1	#0
1611						AOFF		

[Input type] Parameter input
[Data type] Bit path

#2 AOFF When AI contour control mode is off and the parameter of the advanced preview feed forward function is valid, the advanced preview feed forward function is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
1800					FFR			

[Input type] Parameter input
[Data type] Bit path

#3 FFR Feed-forward control in rapid traverse is:
0: Disabled
1: Enabled
Feed-forward is enabled only in normal cutting feed. When this parameter is set to 1, feed-forward is enabled in rapid traverse as well. This capability reduces the servo positional deviation, thus reducing the time required to enter the in-position width at the time of positioning.

NOTE
The parameter setting becomes valid after reference position return is completed.

> **NOTE**
> This parameter is Initial setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

Servo parameter

	#7	#6	#5	#4	#3	#2	#1	#0
2005							**FEEDx**	

[Input type] Parameter input
[Data type] Bit axis

#1 FEEDx Feed forward function is:
0: Disabled.
1: Enabled.

2092	**Advanced preview feed-forward coefficient (ADFF1)**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01%
[Valid data range] 0 to 10000

Specify advanced preview feed-forward coefficient.

	#7	#6	#5	#4	#3	#2	#1	#0
2214				**FFCHG**				

[Input type] Parameter input
[Data type] Bit axis

#4 FFCHG The cutting/rapid feed-forward switching function is:
0: Disabled.
1: Enabled.

> **NOTE**
> When the cutting/rapid feed-forward switching function is used for PMC axis control, set bit 6 (NCI) of parameter No.8004 to 0, and check in-position in the appropriate in-position range.

Signal

Superimposed ahead signal OVLN <Gn531.4>

[Classification] Input signal
[Function] Start superimposed ahead mode.
[Operation] When this signal is 1, the control device operates as follows:
- The corresponding path enters the superimposed ahead mode.

To switch superimposed ahead signal to 1 or 0, it is necessary to stop all axes (except for PMC axes) of path where this signal is switched. When axis moves, alarm DS0071 “START OR RELEASE OF SUPERIMPOSED AHEAD CANNOT BE DONE” is generated. When bit 1 (PFE) of parameter No.8008 is set to 1, change this signal after stopping the PMC axes.

Superimposed ahead under way signal OVLNS <Fn545.1>

[Classification] Output signal

[Function] Indicates that superimposed ahead mode is executing.

[Output cond.] This signal becomes 1 when:

- The corresponding path is executing superimposed ahead mode.

  This signal becomes 0 when:

- The corresponding path is not executing superimposed ahead mode.

Alarm and message

Number	Message	Description
PS0563	CANNOT USE PMC AXIS (ADFF)	Change the related path to superimposed ahead mode when PMC axis control (enabled advanced preview feed forward) is used for the axes related to the following functions. - Composite control - Superimposed control

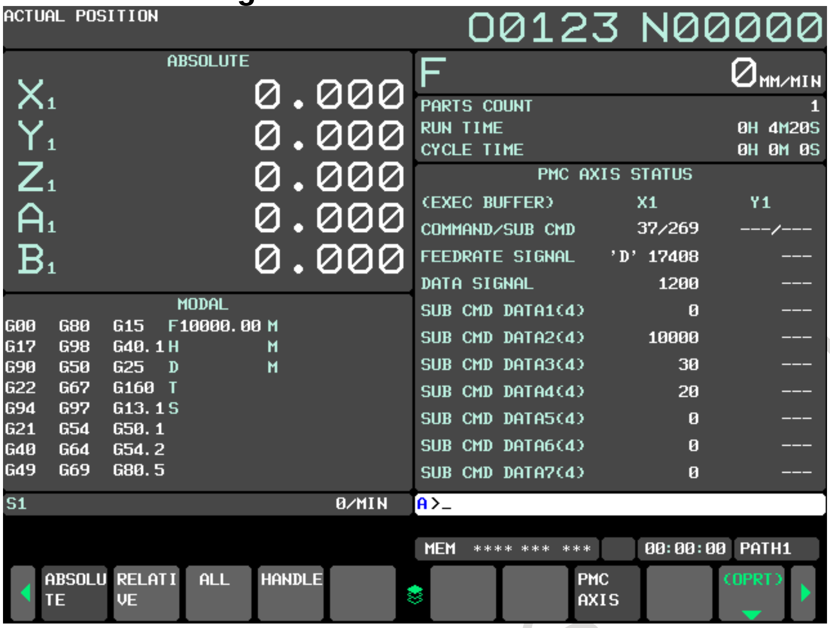
16.1.3 PMC Axis Status Display Function

Overview

This function displays the status of a command specified by PMC axis control and the status of signals related to PMC axis control. This function can be used to easily identify the reason why PMC axis control does not operate normally or to easily check what is specified in PMC axis control in progress.

- PMC axis control signal status screen

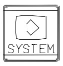
- **PMC axis control executing buffer status screen**



Explanation

- **Screen display procedure**

1 Set the bit 7 (EAC) of parameter No. 3137 =1.

2 Press the function key  .

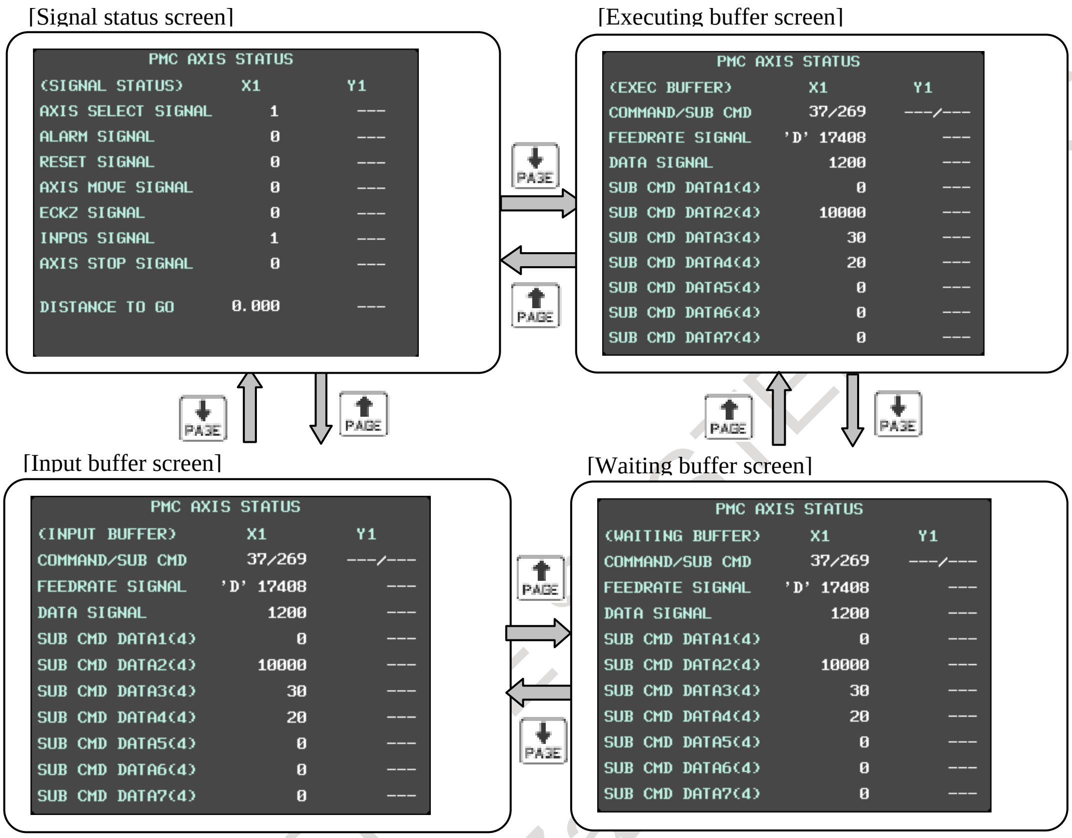
3 Press the continuous menu key several times, then press the chapter selection soft key [PMC AXIS].

4 “Signal status screen for PMC axis control” screen appears.

- Screen configuration and switching to each screen

There are four screens: signal status screen, executing buffer display screen, operation waiting buffer display screen, and input buffer display screen.

Pressing the PAGE key switches between these screens.



- Display operation on a screen

- Switching display axes
  The axis information currently displayed can be switched to the next axis information. Press soft key [AXIS-L NEXT] to switch to the left-hand axis information or press soft key [AXIS-R NEXT] to switch to the right-hand axis information. Each time soft key [AXIS-L NEXT]/[AXIS-R NEXT] is pressed, the axis is changed as shown below.
  First axis -> Second axis -> Third axis -> ...
  The axis information currently displayed can be switched to the previous axis information. Press soft key [AXIS-L PREV] to switch to the left-hand axis information or press soft key [AXIS-R REV] to switch to the right-hand axis information. Each time soft key [AXIS-L PREV]/[AXIS-R PREV] is pressed, the axis is changed reversely as shown below.
  Fourth axis → Third axis → Second axis → ...

- Switching between decimal display and hexadecimal display
  The value of each item can be switched between decimal display and hexadecimal display. Soft key [HEX DISP] appears when the screen is displayed in decimal or soft key [DEC DISP] appears when the screen is displayed in hexadecimal.

- Signal status screen

- Control axis select signal — Displays the status of control axis select signal EAX <Gn136>.
- Alarm signal — Displays the status of alarm signal EIALg.
- Reset signal — Displays the status of reset signal ECLRg.
- Axis move signal — Displays the status of axis moving signal EGENg.
- ECKZ signal — Displays the status of following zero checking signal ECKZg.
- In-position signal — Displays the status of in-position signal EINPg.
- Aixs stop signal — Displays the status of axis control temporary stop signal ESTPg.
- Distance to go — Displays the remaining traverse distance of the command being executed.

NOTE
1. When the control axis select signal is 0, the remaining traverse distance is indicated as "---".
2. When the associated axis is being moved by other than the ladder or macro executor, the remaining traverse distance is indicated as "---".

- Executing buffer display screen

- Command — Displays axis control command signals EC0g to EC6g in the executing buffer.
- Subcommand — Displays a subcommand number in the executing buffer.
- Feedrate signal — Displays axis control feedrate signals EIF0g to EIF5g in the executing buffer.
- Data signal — Displays axis control data signals EID0g to EID31g in the executing buffer.
- Subcommand data 1 (4) — Four-byte information beginning with [subcommand start address + 4] in the executing buffer is displayed.
- Subcommand data 2 (4) — Four-byte information beginning with [subcommand start address + 8] in the executing buffer is displayed.
- Subcommand data 3 (4) — Four-byte information beginning with [subcommand start address + 12] in the executing buffer is displayed.
- Subcommand data 4 (4) — Four-byte information beginning with [subcommand start address + 16] in the executing buffer is displayed.
- Subcommand data 5 (4) — Four-byte information beginning with [subcommand start address + 20] in the executing buffer is displayed.
- Subcommand data 6 (4) — Four-byte information beginning with [subcommand start address + 24] in the executing buffer is displayed.
- Subcommand data 7 (4) — Four-byte information beginning with [subcommand start address + 28] in the executing buffer is displayed.

NOTE
1. What was specified is displayed only when the buffer is valid. When execution is completed or the buffer is invalid, "---" is displayed.
2. If the PMC axis does not move due to an invalid command etc. or an alarm is issued during execution, what was specified is displayed on this screen. This indication is cleared when the ECLRg signal turns on or an emergency stop occurs. After that "---" is displayed.
3. When the associated axis is being moved by other than the ladder or macro executor, "---" is displayed.
4. The speed signal is displayed in decimal with no sign.

- Waiting buffer display screen

- Command — Displays axis control command signals EC0g to EC6g in the waiting buffer.
- Subcommand — Displays a subcommand number in the waiting buffer.
- Feedrate signal — Displays axis control feedrate signals EIF0g to EIF5g in the waiting buffer.
- Data signal — Displays axis control data signals EID0g to EID31g in the waiting buffer.
- Subcommand data 1 (4) — Four-byte information beginning with [subcommand start address + 4] in the waiting buffer is displayed.
- Subcommand data 2 (4) — Four-byte information beginning with [subcommand start address + 8] in the waiting buffer is displayed.
- Subcommand data 3 (4) — Four-byte information beginning with [subcommand start address + 12] in the waiting buffer is displayed.
- Subcommand data 4 (4) — Four-byte information beginning with [subcommand start address + 16] in the waiting buffer is displayed.
- Subcommand data 5 (4) — Four-byte information beginning with [subcommand start address + 20] in the waiting buffer is displayed.
- Subcommand data 6 (4) — Four-byte information beginning with [subcommand start address + 24] in the waiting buffer is displayed.
- Subcommand data 7 (4) — Four-byte information beginning with [subcommand start address + 28] in the waiting buffer is displayed.

NOTE
1. What was specified is displayed only when the buffer is valid. When the buffer is invalid, "---" is displayed.
2. When the associated axis is being moved by other than the ladder or macro executor, "---" is displayed.
3. The speed signal is displayed in decimal with no sign.

- Input buffer display screen

- Command — Displays axis control command signals EC0g to EC6g in the input buffer.
- Subcommand — Displays a subcommand number in the input buffer.
- Feedrate signal — Displays axis control feedrate signals EIF0g to EIF5g in the input buffer.
- Data signal — Displays axis control data signals EID0g to EID31g in the input buffer.
- Subcommand data 1 (4) — Four-byte information beginning with [subcommand start address + 4] in the input buffer is displayed.
- Subcommand data 2 (4) — Four-byte information beginning with [subcommand start address + 8] in the input buffer is displayed.
- Subcommand data 3 (4) — Four-byte information beginning with [subcommand start address + 12] in the input buffer is displayed.
- Subcommand data 4 (4) — Four-byte information beginning with [subcommand start address + 16] in the input buffer is displayed.
- Subcommand data 5 (4) — Four-byte information beginning with [subcommand start address + 20] in the input buffer is displayed.
- Subcommand data 6 (4) — Four-byte information beginning with [subcommand start address + 24] in the input buffer is displayed.
- Subcommand data 7 (4) — Four-byte information beginning with [subcommand start address + 28] in the input buffer is displayed.

NOTE
1 What was specified is displayed only when the buffer is valid. When the buffer is invalid, "---" is displayed.
2 If an alarm occurs due to a command error, what was specified is displayed on this screen. This indication is cleared when the ECLRg signal turns on or an emergency stop occurs. Then, "---" is displayed.
3 When the associated axis is being moved by other than the ladder or macro executor, "---" is displayed.
4 The speed signal is displayed in decimal with no sign.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3137	EAC							

[Input type] Parameter input
[Data type] Bit

#7 EAC The PMC axis status display screen is:
0: Not displayed.
1: Displayed.

16.2 EXTERNAL DATA INPUT

Overview

In the external data input function, the CNC can be operated by the external data which use the signal of PMC.
This function is an optional function.

There are following functions in the external data input.
- External tool offset
- External program number search
- External work coordinate system shift
- External machine zero point shift
- External alarm message
- External operator message
- Substitution of the number of machined parts and number of required parts

NOTE
Ladder sequence of the external data input can be easily incorporated by using PMC function. Functional instruction EXIN(External Data Input: SUB 42) and functional instruction DISPB(Display Message: SUB 41) that can be easily implemented the external data input are provided.
For details on functional instruction EXIN and functional instruction DISPB, refer to the PMC Programming Manual (B-64513EN).

Explanation

- The basic external data input procedure

The following signals in the Table 16.2 (a) are used to send data from the PMC to the CNC.

Table 16.2 (a)

Signal name	Input signal	Output signal
Address signal for external data input	EA0 to EA6	
Data signal for external data input	ED0 to ED31	
Read signal for external data input	ESTB	
Read completion signal for external data input		EREND
Search completion signal for external data input		ESEND

The basic external data input procedure is described below:

(1) The PMC sets the address signals, EA0 to EA6 that indicate the data type and data signals ED0 to ED31.
(2) The PMC sets the read signal ESTB to “1”.
(3) When the ESTB signal is set to “1”, the control unit reads the address.
(4) After reading the address, the control unit sets the read completion signal EREND to “1”.
(5) When the EREND signal is set to “1”, the PMC sets the ESTB signal to “0”.
(6) When the ESTB signal is set to “0”, the control unit sets the EREND signal to “0”. This completes the data input procedure. New data can now be entered.

The timing diagram is shown below (Fig. 16.2 (a)):

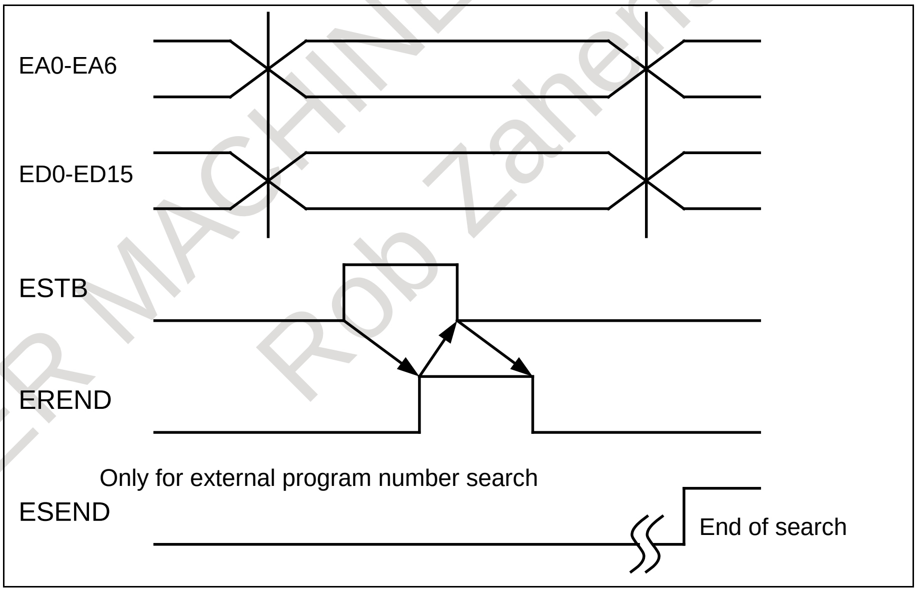


Fig. 16.2 (a) Timing diagram of External data input

Table 16.2 (b)

No.	Item	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0	ED31 to ED16	ED15 to ED0
1	External program number	1	0	0	0	X	X	X	X	Program number (BCD 8 digits with unsign)	
2	External tool offset	1	0	0	1	X	X	X	X	Offset value (BCD 8 digits with sign *1)	
3	Extended external machine zero point shift	1	0	1	0	axis code				Shift value (BCD 8 digits with sign *1)	

No.	Item	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0	ED31 to ED16	ED15 to ED0
4	External machine zero point shift	1	0	1	1	axis code					Extended external machine zero point shift value(binary) -9999 to +9999 *2
5	Alarm set	1	1	0	0	0	0	0	0		Alarm No. (binary) 0 to 999 (0 to 4095)
	Alarm clear	1	1	0	0	0	0	0	1		Alarm No. (binary) 0 to 999 (0 to 4095)
	Operator message set	1	1	0	0	0	1	0	0		message No. (binary) 0 to 999 (0 to 4095)
	Operator message clear	1	1	0	0	0	1	0	1		message No. (binary) 0 to 999 (0 to 4095)
	Message	1	1	0	0	0	X	1	1		character(Character code)
6	Substitute No. of parts required	1	1	1	0	0	0	0	0		No. of parts required (BDC4 digits with unsign)
	Substitute No. of parts machined	1	1	1	0	0	0	0	1		No. of parts machined (BDC4 digits with unsign)

NOTE

1. When bit 3 (EED) of parameter No. 6301 is set to 0, BCD 4 digits are used.
2. By setting bit 7 (EEX) of parameter No. 6300 to 1, a shift value of ±0 to 999999999 can be specified using ED0 to ED31.

NOTE

For axis code, refer to the Table 16.2 (c).

Table 16.2 (c)

Axis	Axis code			
	EA3	EA2	EA1	EA0
1st axis	0	0	0	0
2nd axis	0	0	0	1
3rd axis	0	0	1	0
4th axis	0	0	1	1
5th axis	0	1	0	0
6th axis	0	1	0	1
7th axis	0	1	1	0
8th axis	0	1	1	1
9th axis	1	0	0	0
10th axis	1	0	0	1
11th axis	1	0	1	0
12th axis	1	0	1	1
13th axis	1	1	0	0
14th axis	1	1	0	1
15th axis	1	1	1	0
16th axis	1	1	1	1

- External Program Number Search

A program number (1 to 9999) is specified from an extended source and is selected in the CNC memory.

For machines that can load several kinds of workpieces, this function can automatically select the program to be executed corresponding to a specific workpiece.

Data for the external program number search is accepted regardless of CNC mode, but the search execution can be made only in the reset state in MEM mode.

The ESEND signal switches from “0” to “1” on completion of external program number search. This signal does not turn to “0” unless the cycle start or reset signal is input, or another search is made. Use ESEND to initiate a cycle start signal after the search.

Because a search operation is deferred until a reset occurs in the MEM mode, an external program number search attempted during a CNC operation (the OP signal is “1”) results in the program being started immediately after the end of an automatic operation in a sequence in which the automatic operation is started by checking only the ESEND signal (search completion signal for external data input).

For this reason, using the bit 3 (ESC) of parameter No. 6300 enables an external program number search to be canceled at a CNC reset. Concretely, if a reset signal is input between the time the ESTB signal (read signal for external data input) is input and the time a search would be executed, setting the bit 3 (ESC) of parameter No. 6300 to 1 keeps the search from being executed. At the same time, the controller uses the ESCAN (search cancel signal for external data input) signal to inform the PMC that the search has been canceled.

The ESEND signal (search completion signal for external data input) will not become “1”, because the search is canceled.

The controller checks the state of the RST (NC reset) signal for a reset input between the rising edge of the ESTB signal (read signal for external data input) and the start of the search. Concretely, the controller cancels the external program number search if the RST (NC reset) signal becomes “1” even momentarily in this period. The ESCAN signal (search cancel signal for external data input) becomes “1” at the beginning of the search.

However, it becomes “0” when the next search is executed (for example, when a cycle start or reset signal is input), similarly to the ESEND signal (search completion signal for external data input).

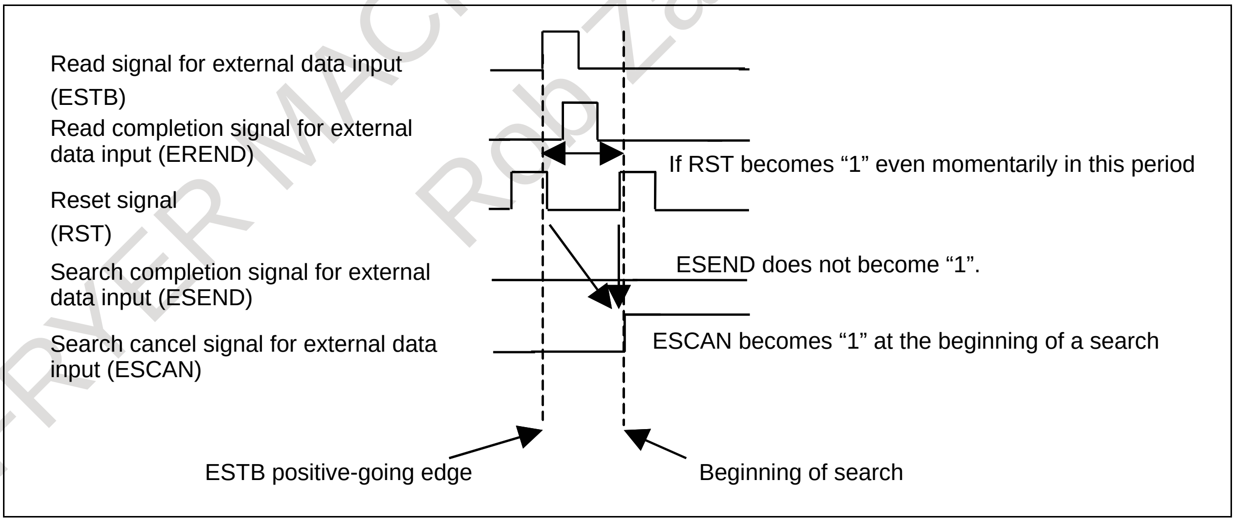


Fig. 16.2 (b) Timing diagram of External data input

This function searches the following folder for a target program:

When parameter No. 3467 is set to 0:

Searches the default foreground folder for the program.

When parameter No. 3467 is set to other than 0:

Searches the initial folder specified in parameter No. 3467 for the program.

NOTE

1. The external program number search is available when bit 4 (ESR) of parameter No. 6300 is set to 1.
2. In reset state the automatic operation lamp is off. If the start button is pushed in the cycle operation stop or hold state, search execution starts from the actual position indicated by the pointer.
3. When there is no program stored in memory corresponding to the set program number, the alarm DS1128 occurs.
4. When the program number search is set to 0, the alarm DS0059 occurs.
5. Data for the external program number search is accepted regardless of the mode, but the search execution can be made only in the reset state. Therefore, in case that the PMC sequence, which the cycle start is executed by checking search completion signal for external data input only, is used, if the external program number search is commanded twice, the program is executed twice.
   (When CNC accepts a command of the program number search, the command is not cancelled even if CNC becomes reset state by external reset signal and so on.) (See Fig. 16.2 (c).)
   If the program execution after reset becomes a problem, make the PMC sequence not to execute the cycle start after reset.
6. Parameter No. 3467 is also used by the “external workpiece number search”.
7. Parameter No. 3467 is also used by the “macro executor program reference and write function”.

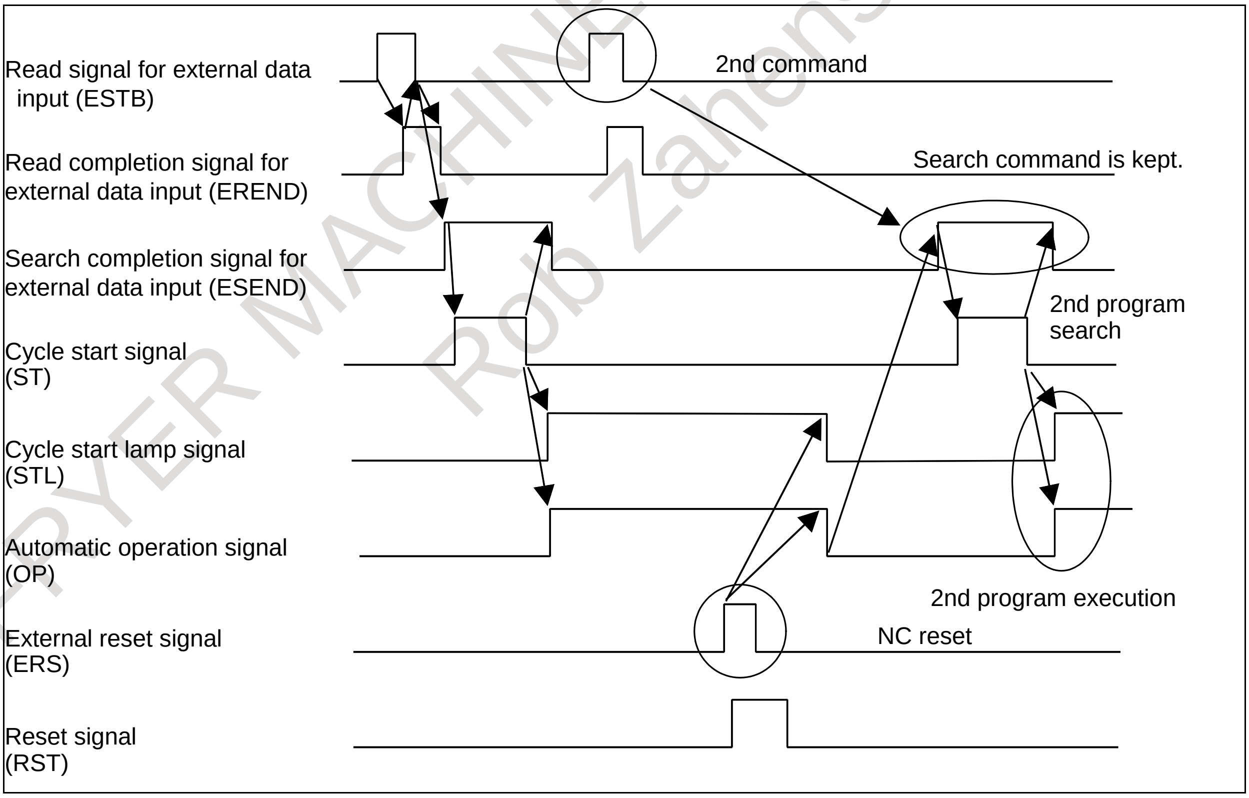


Fig. 16.2 (c) External program search and program execution after NC reset

- External Tool offset

These signals provide for changing the tool compensation value via the PMC. When the offset number is specified by a part program, data input from the PMC is added to the offset value. The offset value can also be used as input data itself by specifying the input signal.

When the machine tool is equipped with automatic tools or workpiece measuring functions, the offset value can be corrected using this function, by inputting the error from the correct value into the CNC via PMC.

If the tool compensation value is externally input when offset number 0 is specified in a part program (an offset cancel) in the lathe turning machine, the workpiece coordinate system shifts by the entered quantity.

The external tool offset range is +/ - 79999999.

> **NOTE**
> 1 When data is out of range, the alarm DS1128 occurs.
> 2 When the data of ineffective function is specified, the alarm DS1121 occurs.
> 3 If bit 3 (EED) of parameter No. 6301 is set to 1, the Data area can be specified by 32bit data specification.
> Under the setting, the external tool offset range is up to +/ - 79999999.

Data specification for external tool offset (For M series)

Address							Data *Note				
EA6	EA5	EA4	EA3	EA2	EA1	EA0	ED31	ED30	. . .	ED1	ED0
0	0	1	a/i	g/w	0	r/l	Sign	Offset value			

Sign: 0: + / 1: -

Offset value: Specified by BCD 8-digit code (0 to +/- 79999999) *Note

a/i	0	Incremental specification
	1	Absolute specification
g/w	0	Specification of tool wear offset value
	1	Specification of tool geometry offset value
r/l	0	Specification of tool length compensation value
	1	Specification of cutter compensation value

*Note: If the bit 3 (EED) of parameter No. 6301 is set to 0, the offset value is specified by using signals ED0 to ED14, the sign is decided by the signal ED15.
The range of data is available to specify the data from 0 to +/- 7999.

Data specification for external tool offset (For T series)

Address							Data *Note				
EA6	EA5	EA4	EA3	EA2	EA1	EA0	ED31	ED30	. . .	ED1	ED0
0	0	1	a/i	g/w	X/Z/R		Sign	Offset value			

Sign: 0: + / 1: - ; Offset value: Specified by BCD 8-digit code (0 to +/- 79999999) *Note

a/i	0	Incremental specification
	1	Absolute specification
g/w	0	Specification of tool wear offset value
	1	Specification of tool geometry offset value
X/Z/R	EA1=0, EA0=0	Specification of X axis offset
	EA1=0, EA0=1	Specification of Z axis offset
	EA1=1, EA0=0	Specification of tool nose radius compensation value

*Note: If the bit 3 (EED) of parameter No.6301 is set to 0, the offset value is specified by using signals ED0 to ED14, the sign is decided by the signal ED15.
The range of data is available to specify the data from 0 to +/- 7999.

NOTE
The Y-axis offset cannot be specified.

- External workpiece coordinate system shift

In the external workpiece coordinate system shift, the shift value can be externally modified by the signal of PMC.
Each axis has this shift value (parameter No. 1220), and this shift value is added to all the workpiece coordinate systems in common.
The shift value is not lost by cutting off the power supply.
The shift value can not only be added to the current work coordinate system shift but also be substituted for the current value.
The value range is 0 to + / - 79999999.
The unit and setting range are the same as the Tool offset.

NOTE
If bit 3 (EED) of parameter No. 6301 is set to 1, the data area can be specified by 32bit data specification.
Under the setting, the external tool compensation range is up to +/ - 79999999.

- External machine zero point shift

The machine coordinate system can be externally shifted by inputting a shift value.
When the shift value is input, compensation is immediately applied to the corresponding axis and the axis moves. The position accuracy can be improved by combining this function with sensors.
The specification to shift the axis is the same as the external workpiece coordinate system shift.
The compensation value is specified in signals ED0 to ED15 using a binary code ranging from 0 to ±9999. This compensation value must be specified in absolute value. The value which the machine actually moves at input is the difference from the previously stored value.
After the Reference point is established, this function becomes effective. The shift amount is memorized though the machine is not moved when the shift amount is input before the Reference point is established. When the Reference point is established afterwards, the machine is moved for the memorized shift amount.

NOTE

If bit 7 (EEX) of parameter No. 6300 is set to 1, the Data area can be specified by 32bit data specification.
Under the setting, the external machine zero point shift range is up to +/ - 999999999.

⚠ CAUTION

When a large value of compensation is applied at one time, an alarm such as "excessive error on stop" may occur. In this case, input the compensation in several smaller increments.

- External message

(a) External alarm message

By sending alarm number from PMC, the CNC is brought to an alarm status; an alarm message is sent to the CNC, and the message is displayed on the screen of the CNC.
Reset of alarm status is also done with external data.
Up to 4 alarm numbers (16 alarm numbers when bit 1 (M16) of parameter No. 11931 is set to 1) and messages can be sent at a same time. A message consisting of up to 63 ASCII characters can be sent for one alarm number.
In case of using a multi-path system, the alarm will be set separately for each path.
Also the alarm is set separately for one.
It is available to select the range of alarm number and the display form by setting the bit 0 (EXA) of parameter No. 6301.

<u>Bit 0 (EXA) of parameter No. 6301 = 0</u>

- Alarm number 0 to 999 can be sent. To distinguish these alarms from other alarms, the CNC displays them by adding 1000 to an alarm number.

<u>Bit 0 (EXA) of parameter No. 6301 = 1</u>

- Alarm number 0 to 4095 can be sent. The CNC displays them with prefix characters “EX” to an alarm number for display.

(b) External operator message

Message to the operator is given by the signal of PMC, and the message is displayed.

The operator messages can be cleared by external data.

Up to 4 message numbers (bit 1 (M16) of parameter No. 11931) can be sent at a same time. Up to 256 characters can be sent in an operator message.

In case of using a multi-path system, the operator message is common resource for plural path of machine group. It is set each machine group. It is available to display the operator message by using the primary path signal of machine group.

It is available to select the range of message number and the display form by setting the bit 1 (EXM) of parameter No. 6301.

Bit 1 (EXM) of parameter No. 6301 = 0

The message numbers 0 to 999 can be sent.

The message numbers 0 to 99 are displayed along with the message.

To distinguish these alarms from other alarms, the CNC displays them by adding 2000 to an alarm number.

When a message from 100 to 999 is displayed, the message number is not displayed; only its text is displayed.

Bit 1 (EXM) of parameter No. 6301 = 1

The message numbers 0 to 4095 can be sent.

The message numbers 0 to 99 are displayed along with the message.

The CNC displays them with prefix characters "EX" to an alarm for display.

When a message number from 100 to 4095 is displayed, the message number is not displayed; only its text is displayed.

Data specification method in external message

Table 16.2 (d)

Item	EA6	EA5	EA4	EA3	EA2	EA1	EA0	ED15 to ED0(binary)
Alarm set	1	0	0	0	0	0	0	Alarm No.
Alarm clear	1	0	0	0	0	0	1	Alarm No.
Operator message list	1	0	0	0	1	0	0	Message No.
Operator message clear	1	0	0	0	1	0	1	Message No.
Message	1	0	0	0	X	1	1	Character(Note)

NOTE

Two characters are sent at a time
(See ISO code given in the table below.)
ED15 to ED8 . . . . Character code in 1st character
ED7 to ED0 . . . . Character code in 2nd character
When sending only one character, fill the second slot with a code smaller than 20 and it will be ignored.

Character code table

				b8	0	0	0	0	1	1	1	1
				b7	0	0	1	1	0	0	1	1
				b6	1	1	0	0	1	1	0	0
				b5	0	1	0	1	0	1	0	1
b4	b3	b2	b1									
0	0	0	0		SP	0	@	P	~	ー	タ	ミ
0	0	0	1		!	1	A	Q	。	ア	チ	ム
0	0	1	0		"	2	B	R	「	イ	ツ	メ
0	0	1	1		#	3	C	S	」	ウ	テ	モ
0	1	0	0		$	4	D	T	、	エ	ト	ヤ
0	1	0	1		%	5	E	U	・	オ	ナ	ユ
0	1	1	0		&	6	F	V	ヲ	カ	ニ	ヨ
0	1	1	1		'	7	G	W	ァ	キ	ヌ	ラ
1	0	0	0		(	8	H	X	ィ	ク	ネ	リ
1	0	0	1		)	9	I	Y	ゥ	ケ	ノ	ル
1	0	1	0		*	:	J	Z	ェ	コ	ハ	レ
1	0	1	1		+	;	K	[	ォ	サ	ヒ	ロ
1	1	0	0		,	<	L	¥	ャ	シ	フ	ワ
1	1	0	1		-	=	M	]	ュ	ス	ヘ	ン
1	1	1	0		.	>	N	^	ョ	セ	ホ	゛
1	1	1	1		/	?	O	_	ッ	ソ	マ	゜

SP:Space code

Fig. 16.2 (d)

- Substitution of the number of required parts and number of machined parts

Substitution is possible for the number of parts required and the number of parts machined.

Data specification for No. of parts required and No. of parts machined.

Address							Data		
EA6	EA5	EA4	EA3	EA2	EA1	EA0	ED15	....	ED0
1	1	0	0	0	0	0/1	MSB Value of parts no.		LSB

(1) (2) (3)

(1) Set to 110000
(2) 0: No. of parts required
    1: No. of parts machined
(3) BCD 4-digit code (0000 to 9999)

Signals

Address signals for external data input EA0 to EA6 <Gn002.6 – Gn002.0>

[Classification] Input signal

[Function] These signals indicate the type of the entered data.

Data signals for external data input ED0 to ED31 <Gn211, Gn210,Gn000, Gn001>

[Classification] Input signal

[Function] These signals indicate the entered data.
The use of the 32 code signals (16 code signals) varies with the data type.

Read signal for external data input ESTB <Gn002.7>

[Classification] Input signal

[Function] The signal reports that the address and data are set in external data input.
When the signal is set to "1", the control unit reads the address and data for external data input.

Read completion signal for external data input EREND <Fn060.0>

[Classification] Output signal

[Function] This signal reports that the control unit has finished reading the entered data.

Search completion signal for external data input ESEND <Fn060.1>

[Classification] Output signal

[Function] This signal report that program number search, specified by external data input, has been completed.

[Output cond.] This signal is set to "1" when:

- The program number search specified by external data input is completed.

The signal is set to "0" when:

- An automatic operation is started.
- A reset occurs.

Search cancel signal for external data input ESCAN<Fn060.2>

[Classification] Output signal

[Function] This signal notifies PMC of the cancellation of a program number search.

[Output cond.] When a reset is input between the time read signal for external data input ESTB is input and the time a search is executed in the external program number search function, if bit 3 (ESC) of parameter No. 6300 is 1, the search is not executed. At this time, the control unit sets search cancel signal for external data input ESCAN to "1" instead of setting search completion signal for external data input ESEND to "1".

NOTE
This signal is enabled when bit 3 (ESC) of parameter No. 6300 is 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn000	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
Gn001	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
Gn002	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0
Gn210	ED23	ED22	ED21	ED20	ED19	ED18	ED17	ED16
Gn211	ED31	ED30	ED29	ED28	ED27	ED26	ED25	ED24

	#7	#6	#5	#4	#3	#2	#1	#0
Fn060						ESCAN	ESEND	EREND

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6300	EEX			ESR	ESC			

[Input type] Parameter input
[Data type] Bit path

#3 ESC When a reset is input between the input of the external data input read signal ESTB and the execution of a search, the external program number search function:
0: Performs a search.
1: Does not perform a search.

#4 ESR The external program number search function is:
0: Disabled.
1: Enabled.

#7 EEX PMC EXIN function
0: Conventional specifications
1: Extended specifications
If you want to use external machine coordinate system shift which handles ±10.000 or more shift unavailable with the PMC/EXIN command in the conventional specifications, set 1.
When this function is used for a multi-path system, the setting for path 1 is used.
For details of EXIN and how to change ladder software, refer to the PMC manuals.

	#7	#6	#5	#4	#3	#2	#1	#0
6301					EED	NNO	EXM	EXA

[Input type] Parameter input
[Data type] Bit machine group

#0 EXA This bit selects an external alarm message specification.
0: A message number from 0 to 999 can be sent. When displaying an alarm number, the CNC prefixes the character string "EX" to the alarm number obtained by adding 1000 to the message number.
1: A message number from 0 to 4095 can be sent. The CNC prefixes the character string "EX" to a alarm number for display.

#1 EXM This bit selects an external operator message specification.
0: A message number from 0 to 999 can be sent. The message of a message number from 0 to 99 is displayed together with its number. The CNC adds 2000 to a number for distinction. A message number from 100 to 999 is not displayed on the screen, but only the corresponding message is displayed on the screen.
1: A message number from 0 to 4095 can be sent. The message of a message number from 0 to 99 is displayed together with its number. The CNC prefixes the character string "EX" to a message number for display. A message number from 100 to 4095 is not displayed on the screen, but only the corresponding message is displayed on the screen.

#2 NNO When operator messages are set by external data input, a new line operation between one message set with a number and another message set with a different number is:
0: Performed.
1: Not performed.

#3 EED To specify data for external tool compensation and external workpiece coordinate system shift, use:
0: Signals ED15 to ED0.
(The value which can be specified for tool compensation and workpiece coordinate system shift is from 0 to ±7999.)
1: Signals ED31 to ED0.
(The value which can be specified for tool compensation and workpiece coordinate system shift is from 0 to ±79999999.)

6310	Setting for number addition to external operator messages

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word machine group
[Valid data range] 0 to 4096
This parameter sets the number of messages to which message numbers are to be prefixed in external operator message display.
When 0 is set, the same operation as when 100 is set is performed.
[Example] When 500 is set in this parameter, the messages of message numbers 0 to 499 are displayed together with their numbers on the screen. A message number of 500 and up is not displayed on the screen, but only the corresponding message is displayed on the screen.

	#7	#6	#5	#4	#3	#2	#1	#0
11931							M16	

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 M16 In the external data input and external messages, the maximum number of external alarm messages and external operator messages that can be displayed is:
0: 4.
1: 16.

3467	Selection of the target folder among initial folders

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 7, 11 to 20
This parameter selects a folder to be used for the external workpiece number search function, external program number search function, and macro executor program reference and write function among the following initial folders:
0: No specification
1: Root folder (//CNC_MEM)
2: System folder (//CNC_MEM/SYSTEM)
3: MTB-dedicated folder 1 (//CNC_MEM/MTB1)
4: MTB-dedicated folder 2 (//CNC_MEM/MTB2)

5: User folder (//CNC_MEM/USER)
6: Path folder (//CNC_MEM/USER/PATHn) (Note)
(NOTE) PATHn: n is the selected path number (1 to the maximum number of paths).
7: Common program folder (//CNC_MEM/USER/LIBRARY)
8 to 10: Not specified.
11: Path 1 folder (//CNC_MEM/USER/PATH1)
12: Path 2 folder (//CNC_MEM/USER/PATH2)
13: Path 3 folder (//CNC_MEM/USER/PATH3)
14: Path 4 folder (//CNC_MEM/USER/PATH4)

When “0: No specification” is selected, the following folder is used for each function:

- External program number search function
- External workpiece number search function
  (Default foreground folder)
- Macro executor program reference and write function
  (Default background folder)

NOTE
Any user-created folder cannot be specified.
For example, assume that a user folder named PATH3 is created in //CNC_MEM/USER in a 2-path system. User-created folder //CNC_MEM/USER/PATH3 cannot be specified by specifying 13 in this parameter.

Alarm and message

Number	Message	Description
DS0059	SPECIFIED NUMBER NOT FOUND	The No. specified for a program No. or sequence No. search could not be found. There was an I/O request issued for a pot No. or offset (tool data), but either no tool numbers have been input since power ON or there is no data for the entered tool No.
DS0131	TOO MANY MESSAGE	An attempt was made to display an external operator message or external alarm message, but five or more displays were required simultaneously.
DS0132	MESSAGE NUMBER NOT FOUND	An attempt to cancel an external operator message or external alarm message failed because the specified message number was not found.
DS0133	TOO LARGE NUMBER	A value other than 0 to 4095 was specified as the external operator message or the external alarm message number.
DS1120	UNASSIGNED ADDRESS (HIGH)	The upper 4 bits (EIA4 to EIA7) of an external data I/O interface address signal are set to an undefined address (high bits).
DS1121	UNASSIGNED ADDRESS (LOW)	The lower 4 bits (EIA0 to EIA3) of an external data I/O interface address signal are set to an undefined address (low bits).
DS1128	DI.EIDLL OUT OF RANGE	The numerical value input by external data input signals EID0 to EID31 has exceeded the permissible range.
DS1130	SEARCH REQUEST NOT ACCEPTED	No requests can be accepted for a program No. or a sequence No. search as the system is not in the memory mode or the reset state.
DS1131	EXT-DATA ERROR (OTHER)	[External Data I/O] An attempt was made to input tool data for tool offset by a tool No. during loading by the G10 code.

16.3 EXTENDED EXTERNAL MACHINE ZERO POINT SHIFT

Overview

The conventional external machine zero point shift value function cannot make shifts on multiple axes simultaneously.
With this extended function, external machine zero point shifts can be performed on all controlled axes.
An external machine zero point shift value is to be set in a parameter-set R area. A shift value must be specified using a binary code, and the absolute value of a number from -32767 to 32767 must be specified.

Explanation

- Setting

This function is enabled by setting bit 0 (EMS) of parameter No. 1203 to 1. Enter a desired external machine zero point shift value at the location starting at the address (R area of the PMC) set in parameter No. 1280.
If 100 is set in parameter No. 1280, enter shift values at the location starting at R100 of the PMC. Set an even number in parameter No. 1280.

Example) When 100 is set in parameter No. 1280

R0100	External machine zero point shift value for the 1st axis (Low)
R0101	External machine zero point shift value for the 1st axis (High)
R0102	External machine zero point shift value for the 2nd axis (Low)
R0103	External machine zero point shift value for the 2nd axis (High)
:	:
R(0100+2(n-1))	External machine zero point shift value for the n-th axis (Low)
R(0100+2(n-1)+1)	External machine zero point shift value for the n-th axis (High)

- Shift value

Specify a shift value by using a two-byte binary code for each axis.
A value from -32767 to 32767 can be specified.
A shift value is assumed to be specified as an absolute value.
The unit is the detection unit.

Example)

Suppose that the incremental system is IS-B, the machine is a millimeter machine (bit 0 of parameter No. 1001 = 0),
the detection unit is 0.0002 mm (CMR (parameter No. 1820) = 10),
and parameter No. 1280 is set to 100.
When the following values are written to the R area :
R102 = 11001100 (CCh)
R103 = 11101101 (EDh)
the machine position on the second axis is shifted, and the shift value at that time is:

EDCCh [pulse] * 0.0002 [mm/pulse] = -0.932 mm

- Relationship with the error compensation functions

This function is superposed on error compensation functions such as the pitch error compensation function and straightness compensation function.

- Coordinate system

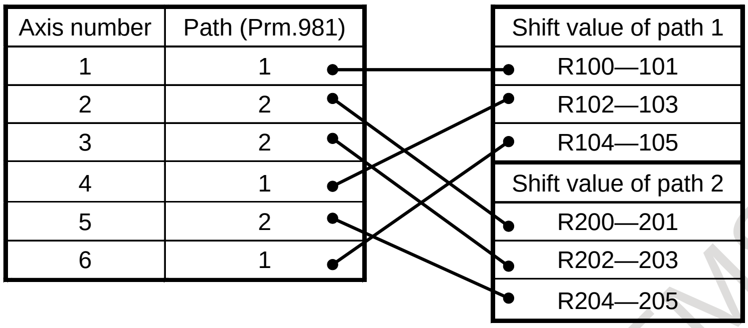
If the machine is moved by this function, the coordinates are not updated.

- Multi-path control

In multi-path control, the locations of the R area are used in ascending order of axis numbers in the paths as shown in the example below.

Example)

When, in parameter No. 1280, 100 is set for path 1 and 200 is set for path 2

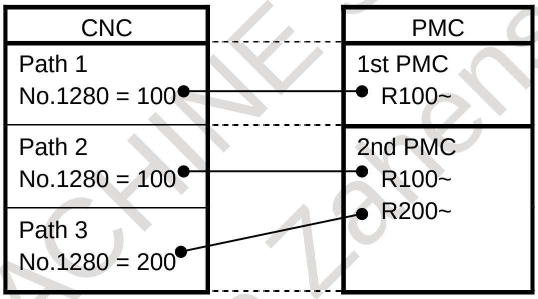


- Multi-path PMC

With a multi-path PMC, the PMC R area allocated to each path is used.

Example)

When the first PMC is assigned to path 1, and the second PMC is assigned to paths 2 and 3, the R area of the first PMC is assigned to shift values for path 1, and the R area of the second PMC is assigned to shift values for paths 2 and 3.



Signal

Extended external machine zero point shift signal
EMZ0 to EMZ15 <Rn to Rn+2*controlled axis count-1>

[Classification] Input signal

[Function] Sets an external machine zero point shift value.

[Operation] When an external machine zero point shift value is set with this signal, the machine position is shifted by the specified value.
A set value is regarded as an absolute value.
As a signal address, specify an arbitrary R address with parameter No. 1280.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Rn	EMZ7	EMZ6	EMZ5	EMZ4	EMZ3	EMZ2	EMZ1	EMZ0
Rn+1	EMZ15	EMZ14	EMZ13	EMZ12	EMZ11	EMZ10	EMZ9	EMZ8
:								
Rn+2*controlled axis count-1	EMZ15	EMZ14	EMZ13	EMZ12	EMZ11	EMZ10	EMZ9	EMZ8

n represents the value set in parameter No. 1280.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1203								EMS

[Input type] Parameter input
[Data type] Bit path

#0 EMS The extended external machine zero point shift function is:
0: Disabled.
1: Enabled.

> **NOTE**
> 1 To use the extended external machine zero point shift function, the external machine zero point shift function or the external data input function is required.
> 2 When the extended external machine zero point shift function is enabled, the conventional external machine zero point shift function is disabled.

> ⚠ **WARNING**
> Set an appropriate value to parameter No.1280 beforehand when you set 1 to this parameter.
> If an internal relay of the set address in parameter No.1280 is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used internal relay.

1280	Start address of signals used with the extended external machine zero point shift function

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] Even number from 0 to 59999

Set the start address of signals used with the extended external machine zero point shift function. If a nonexistent address value is specified, this function is disabled.
If 100 is set, for example, this function uses R100 and up. The last R address to be used depends on the number of controlled axes. When eight controlled axes are used, R100 to R115 are used.

> **NOTE**
> 1 If a nonexistent R address or an address in the system area is set, this function is disabled.
> 2 Set an even number in this parameter.
> 3 When this parameter is 0, an internal relay from address R0 is used.

> ⚠ **WARNING**
> If an internal relay of the set address in this parameter is used by another usage, it may cause an unexpected machine behavior. Be careful enough that you must prevent the competition of the used internal relay.

Warning and Note

> ⚠ **WARNING**
> 1 When writing an external machine zero point shift value to the R area, write a shift value for one axis as a word. If data for one axis is written in two operations, one byte in one operation, an incorrect shift operation can result.
> 2 When writing data with the PMC or FOCAS2 function, provide an interval of about 8 ms after one write operation before starting the next write operation. If this write timing is not observed, an incorrect shift operation can result.

> **NOTE**
> This function can not use together with flexible path axis assignment function.

16.4 IMPROVEMENT OF EXTERNAL MACHINE ZERO POINT SHIFT

Overview

In case of the external machine zero point shift or the extended external machine zero point shift, the maximum shift value per one execution cycle is extended.

Explanation

In case of the external machine zero point shift or the extended external machine zero point shift, when all of following conditions are satisfied, the maximum shift value per one execution cycle is multiplied by 10. (The positive side is 1270. The negative side is -1280.)

- Bit 1 (EMF) of parameter No. 1814 = 1
- Increment system is B or C.
- Bit 2 (HPE) of parameter No. 1816 = 1

> **NOTE**
> 1 In case of the compensation functions, when this function is valid, the maximum compensation value per one execution cycle is multiplied by 10. (The positive side is 1270. The negative side is -1280.)
> - Stored pitch error compensation
> - Bi-directional pitch error compensation
> - Interpolation type pitch error compensation
> - Smooth backlash compensation
>
> 2 Bit 1 (EMF) of parameter No.1814 is invalid on the following axes.
> - Axis for Cs contour control
> - Axis for Spindle positioning

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1814							EMF	

[Input type] Parameter input
[Data type] Bit axis

#1 EMF In case of the external machine zero point shift or the extended external machine zero point shift, the maximum shift value per one execution cycle is:
0: Conventional specification. (The positive side is 127. The negative side is -128.)
1: Multiplied by 10. (The positive side is 1270. The negative side is -1280.)
When all of following conditions are satisfied, this parameter is valid.
- Increment system is B or C.
- Bit 2 (HPE) of parameter No. 1816 = 1

NOTE
1 In case of the compensation functions, when this parameter is valid, the maximum compensation value per one execution cycle is multiplied by 10. (The positive side is 1270. The negative side is -1280.)
- Stored pitch error compensation
- Bi-directional pitch error compensation
- Interpolation type pitch error compensation
- Smooth backlash compensation
2 This parameter is invalid on the following axes.
- Axis for Cs contour control
- Axis for Spindle positioning

	#7	#6	#5	#4	#3	#2	#1	#0
1013							ISCx	ISAx

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ISAx
#1 ISCx Increment system of each axis

Increment system	Bit 1 (ISC)	Bit 0 (ISA)
IS-A	0	1
IS-B	0	0
IS-C	1	0

	#7	#6	#5	#4	#3	#2	#1	#0
1816						HPE		

[Input type] Parameter input
[Data type] Bit axis

#2 HPE The compensation amount by the error compensation can be output based on:
0: Detection unit
1: 1/1000 of the detection unit (same unit as for nano compensation)

NOTE
1 This parameter is valid in the following error compensation.
- Smooth backlash compensation
- Interpolation type straightness compensation
- Interpolation type straightness compensation 3072 points
- Interpolation type pitch error compensation

2 In error compensation, set compensation data such as pitch error data by the detection unit regardless of the setting of this parameter.
3 This parameter is valid on the following axes.
- Servo axis
- Axis for which Cs contour control or Spindle positioning is enabled

4 When this parameter is 1, the compensation amount by following error compensation can be output based on the detection unit.
- Backlash compensation
- Stored pitch error compensation
- Straightness compensation
- Inclination compensation
- Bi-directional pitch error compensation

16.5 EXTERNAL WORKPIECE NUMBER SEARCH

Overview

A machining program stored in the part program memory can be started by an external signal.
When automatic operation is started in the memory operation mode during the reset state, the program is searched for from a specified workpiece number and then executed from the beginning.
For a machine with a function for automatically loading several types of workpieces, this function can be used to automatically select and execute the program according to the workpiece.
This function searches the following folder for a target program:
When parameter No. 3467 is set to 0:
Searches the default foreground folder for the program.
When parameter No. 3467 is set to other than 0:
Searches the initial folder specified in parameter No. 3467 for the program.

> **NOTE**
> 1 Parameter No. 3467 is also used by the “external program number search”.
> 2 Parameter No. 3467 is also used by the “macro executor program reference and write function”.

Signal

External workpiece number search signals
PN1,PN2,PN4,PN8,PN16 <Gn009.0 to Gn009.4>

[Classification] Input signal

[Function] These signals specify a workpiece number to be executed in the memory operation mode. These are 5-bit code signals and correspond to the workpiece numbers as shown Table 16.5 (a) (binary codes).

Table 16.5 (a)

External workpiece number search signals					Workpiece number
PN16	PN8	PN4	PN2	PN1	
0	0	0	0	0	00
0	0	0	0	1	01
0	0	0	1	0	02
Omission					
1	1	1	1	0	30
1	1	1	1	1	31

Of these numbers, workpiece number 00 has special meaning "making no search". Therefore, a number from 01 to 31 can be specified as a workpiece number.

> **NOTE**
> The signals are also used to specify a file number for finding the beginning of the file in inputting an external program.

[Operation] Searches the program number corresponding to the workpiece number specified by these signals when:
Automatic operation is started (the automatic operation start signal (ST) changes from “1” to “0”) in the memory operation mode if automatic operation is in the reset state (the automatic operation signal (OP) is “0”).
When the workpiece number is 00, however, no search is carried out.
The searched program is used in:
- Automatic operation in the memory operation mode
- Foreground edit in the memory edit mode

Extended external workpiece number search signals
EPN0 to EPN13 <Gn024.0 to Gn025.5>

[Classification] Input signal

[Function] These signals specify a workpiece number to be executed in the memory operation mode. The correspondence with the workpiece numbers is shown Table 16.5 (b) (binary codes).

Table 16.5 (b)

Extended external workpiece number search signals														Workpiece number
EPN13	EPN12	EPN11	EPN10	EPN9	EPN8	EPN7	EPN6	EPN5	EPN4	EPN3	EPN2	EPN1	EPN0	
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0000
0	0	0	0	0	0	0	0	0	0	0	0	0	1	0001
0	0	0	0	0	0	0	0	0	0	0	0	1	0	0002
Omission														
1	0	0	1	1	1	0	0	0	0	1	1	1	0	9998
1	0	0	1	1	1	0	0	0	0	1	1	1	1	9999

Of these numbers, workpiece number 0000 has special meaning "making no search". Therefore, a number from 0001 to 9999 can be specified as a workpiece number.
These signals are used in place of the external workpiece number search signals (PN1 to PN16) when the signal is selected by bit 1 (EPN) of parameter No. 3006.

[Operation] Operation of these signals is same as operation of external workpiece number search signals.

External workpiece number search start signal EPNS <Gn025.7>

[Classification] Input signal

[Function] This signal executes only the search function of the workpiece number search and does not perform automatic operation. When this signal changes from ”1” to ”0”, the search function is executed.
When bit 2 (EPS) of parameter No. 3006 is set to 1, this signal is enabled and the search function by ST is disabled.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn009				PN16	PN8	PN4	PN2	PN1
Gn024	EPN7	EPN6	EPN5	EPN4	EPN3	EPN2	EPN1	EPN0
Gn025	EPNS		EPN13	EPN12	EPN11	EPN10	EPN9	EPN8

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3006						EPS	EPN	

[Input type] Parameter input
[Data type] Bit

#1 EPN As signals for specifying workpiece numbers for external workpiece number search:
0: The external workpiece search signals (PN1 to PN16) are used. (A number from 1 to 31 can be specified.)
1: The extended external workpiece number search signals (EPN0 to EPN13) are used. (A number from 1 to 9999 can be specified.)

#2 EPS As the signal for starting external workpiece number search:

0: The automatic operation start signal ST is used. When automatic operation (memory operation) is started, a search is made.

1: The external workpiece number search start signal EPNS is used. ST does not start a search.

3020	Correspondence between workpiece numbers and program numbers in external workpiece number search (PN)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] -1 to 99

This parameter has the following meaning according to the value set.

- When a value from 0 to 99 is set
  (Program number) = (setting)*100+(workpiece number)
  This means that the setting specifies the higher 2 digits of a program number.
- When the value -1 is set
  The higher 2 digits of a program number represent the minimum of the existing program numbers.

<u>Example</u>

When workpiece number 21 is specified, program numbers such as O0021, O0121, and O0221 are searched for. If O0021 is not found, but O0121 and O0221 are found, O0121 is selected as the program number.

> **NOTE**
> This parameter is valid when a workpiece number is specified using the PN1 to PN16 signals (when bit 1 (EPN) of parameter No. 3006 is set to 0).

3467	Selection of the target folder among initial folders

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 7, 11 to 20

This parameter selects a folder to be used for the external workpiece number search function, external program number search function, and macro executor program reference and write function among the following initial folders:

0:	No specification	
1:	Root folder	(//CNC_MEM)
2:	System folder	(//CNC_MEM/SYSTEM)
3:	MTB-dedicated folder 1	(//CNC_MEM/MTB1)
4:	MTB-dedicated folder 2	(//CNC_MEM/MTB2)
5:	User folder	(//CNC_MEM/USER)
6:	Path folder	(//CNC_MEM/USER/PATHn) (Note)
	(NOTE) PATHn: n is the selected path number (1 to the maximum number of paths).	
7:	Common program folder	(//CNC_MEM/USER/LIBRARY)
8 to 10:	Not specified.	
11:	Path 1 folder	(//CNC_MEM/USER/PATH1)
12:	Path 2 folder	(//CNC_MEM/USER/PATH2)
13:	Path 3 folder	(//CNC_MEM/USER/PATH3)
14:	Path 4 folder	(//CNC_MEM/USER/PATH4)

When “0: No specification” is selected, the following folder is used for each function:

- External program number search function
- External workpiece number search function
  (Default foreground folder)
- Macro executor program reference and write function
  (Default background folder)

NOTE
Any user-created folder cannot be specified.
For example, assume that a user folder named PATH3 is created in //CNC_MEM/USER in a 2-path system. User-created folder //CNC_MEM/USER/PATH3 cannot be specified by specifying 13 in this parameter.

Alarm and message

Number	Message	Description
DS0059	SPECIFIED NUMBER NOT FOUND	The program corresponding to the specified workpiece No. could not be found.

16.6 EXTERNAL KEY INPUT

Overview

MDI key codes can be sent from the PMC to CNC by means of interface signals. This allows the CNC to be controlled in the same way as when the operator performs MDI key operation.

Signal

- Signal list

Control is realized by exchanging the following interface signals (Table 16.6 (a)) between the PMC and CNC:

Table 16.6 (a)

Signal name	Abbreviation
External key input mode selection signal (input)	ENBKY
Key code signals (input)	EKC0 to EKC7
Key code read signal (input)	EKSET
Key code read completion signal (output)	EKENB
Key input disable signal (output)	INHKY
Program screen display mode signal (output)	PRGDPL

- Signal detail

The processing flow in the PMC is shown Fig. 16.6 (a).

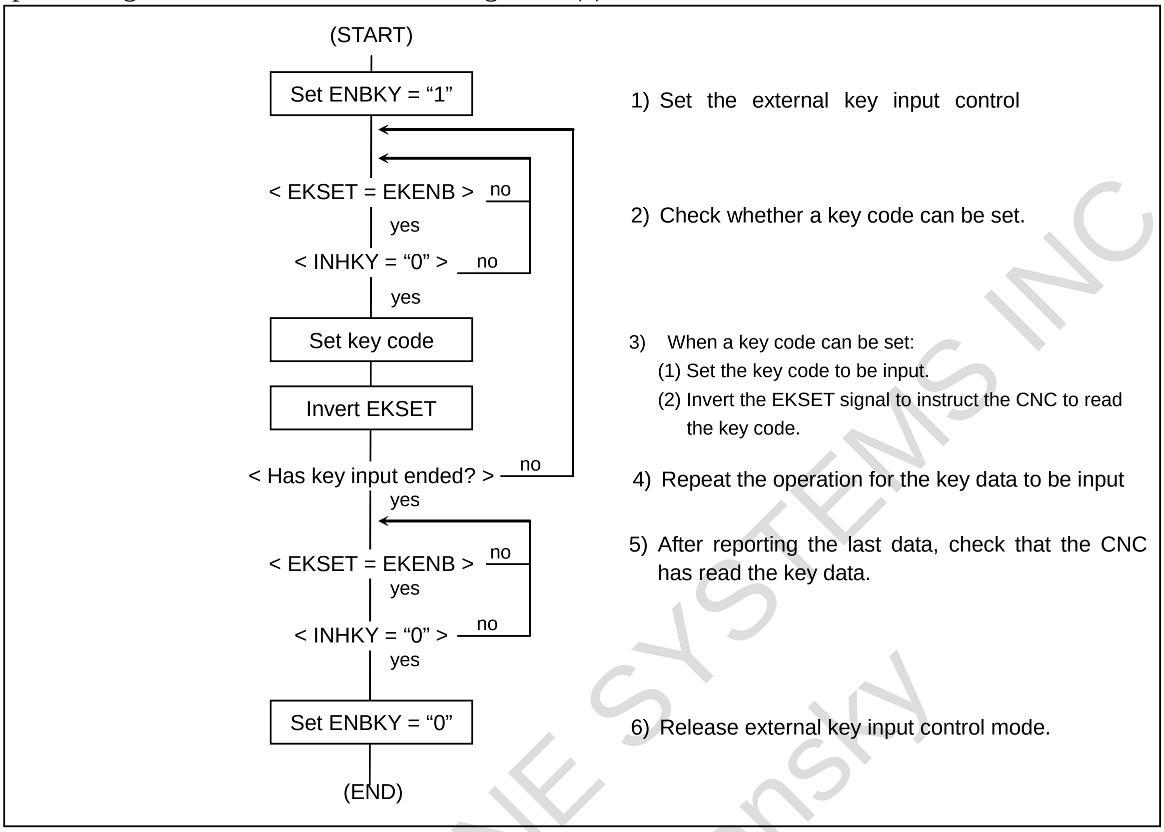


Fig. 16.6 (a)

> **NOTE**
> Read processing is controlled by exclusive-ORing (XOR) the key code read signal (EKSET) with the read completion signal (EKENB). When the EKSET and EKENB signals differ in their logic, the CNC reads the input key code. Once reading has been completed, the CNC inverts the EKENB signal to match its logic with that of the EKSET signal.
> In the PMC, on the other hand, a new key code cannot be set while the EKSET and EKENB signals differ in their logic.

External key input mode selection signal ENBKY<G066.1>

[Classification] Input signal

[Function] While this signal is turned on "1", external key input control is enabled. In external key input control mode, any MDI key operations are ignored.

> **NOTE**
> 1 When ENBKY is set to "1" to enable external key input control, key code input based on the C Language Executor is also ignored.
> 2 Key codes input with the external key input function can be read by the macro executor and C Language Executor.

NOTE

3 Even when external key input control is enabled, pressing the SHIFT key on the MDI keyboard for 5 seconds can produce a hard copy of the screen.

Pressing the CAN key while a hard copy of the screen is being produced can cancel the production of a hard copy of the screen.

Even if the key code of the CAN key is input with the external key input function, the production of a hard copy of the screen cannot be stopped.

Key code read signal EKSET<G066.7>

[Classification] Input signal

[Function] This signal instructs the CNC to read the input key code.
If the logical state of this signal is opposite to the logical sate of the key code read completion signal EKENB, a key code read is specified.

Key code signals EKC0 to EKC7<G098>

[Classification] Input signal

[Function] These signals set an input key code.

Key input disable signal INHKY<F053.0>

[Classification] Output signal

[Function] While this signal is “1”, no key code is accepted in external key input control mode.

[Output cond.] This signal is set to “1” during program editing or program input/output.

Program screen display mode signal PRGDPL<F053.1>

[Classification] Output signal

[Function] This signal posts whether the CNC is displaying the program screen.

[Output cond.] This signal is set to “1” when the program screen is displayed.

Key code read completion signal EKENB<F053.7>

[Classification] Output signal

[Function] This signal reports that the CNC has read a key code.

[Output cond.] When the CNC completes key code read operation, the logical state of this signal is set to the same state as for the key code read signal EKSET.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G066	EKSET						ENBKY	
G098	EKC7	EKC6	EKC5	EKC4	EKC3	EKC2	EKC1	EKC0

	#7	#6	#5	#4	#3	#2	#1	#0
F053	EKENB						PRGDPL	INHKY

MDI key code

Codes in the table are given in hexadecimal. For example,

A corresponds to 41H in hexadecimal.

corresponds to 90H in hexadecimal.

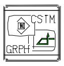
- MDI Key Code Table (00H to 7FH)

	0	1	2	3	4	5	6	7
0			Space	0	@	P		
1			!	1	A	Q		
2			“ (Double-quote)	2	B	R		
3			#	3	C	S		
4			$	4	D	T		
5				5	E	U		
6			&	6	F	V		
7			‘ (Single-quote)	7	G	W		
8			(	8	H	X		
9	TAB		)	9	I	Y		
A	; (EOB)		*	: (Colon)	J	Z		
B			+	; (Semicolon)	K	[		
C			, (Comma)	<	L	¥		
D			- (Minus)	=	M	]		
E			. (Period)	>	N			~
F			/	?	O	_ (Underline)		

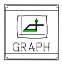
- MDI Key Code Table (80H to FFH)

	8	9	A	B	C	D	E	F
0		RESET	[VF9] (Note 3)					[F0] (Note 2)
1			[VF8] (Note 3)					[F1] (Note 2)
2			[VF7] (Note 3)					[F2] (Note 2)
3			[VF6] (Note 3)					[F3] (Note 2)
4	SHIFT	INSERT	[VF5] (Note 3)				AUX	[F4] (Note 2)
5		DELETE	[VF4] (Note 3)					[F5] (Note 2)
6	CAN	ALTER	[VF3] (Note 3)					[F6] (Note 2)
7		ALT	[VF2] (Note 3)					[F7] (Note 2)
8	Cursor →	INPUT	[VF1] (Note 3)				POS	[F8] (Note 2)
9	Cursor ←						PROG	[F9] (Note 2)
A	Cursor ↓	HELP					OFFSET SETTING	
B	Cursor ↑	CTRL					SYSTEM	
C		ABC/abc					MESSAGE	
D							GRAPH (CUSTOM) (Note 1)	
E	PAGE ↓						CUSTOM (Note 1)	[FR] (Note 2)
F	PAGE ↑						CUSTOM2	[FL] (Note 2)

NOTE

1 For the small keyboard, 0EDH is assigned to 

.

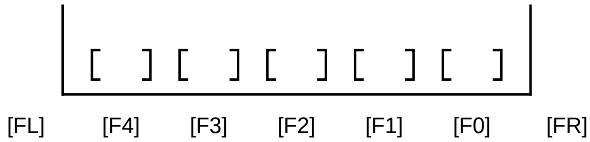
For the standard keyboard, 0EDH is assigned to 

. 0EEH is assigned to 

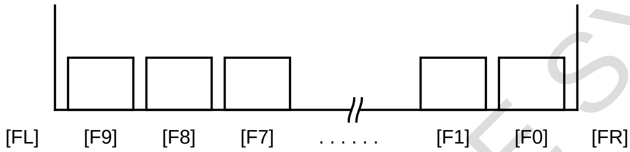
.

2 Handling of the soft keys
[F0] to [F9], [FR], and [FL] in the key code table are the key codes for the soft keys. They are associated with the MDI keys as shown below.

Key configuration for 7-soft key type LCD, or etc. : 5 keys + 2 keys ([F0] to [F4] and [FR], [FL])



Key configuration for 12-soft key type LCD, or etc.: 10 keys + 2 keys ([F0] to [F9] and [FR], [FL])



3 Handling of the vertical soft keys
[VF1] to [VF9] in the key code table are the key codes for the vertical soft keys. From the top to the bottom, [VF1] to [VF9] are arranged.

16.7 ONE TOUCH MACRO CALL

Overview

This function enables the following three operations in pushing the switch installed in the machine only by the change in a minimum LADDER program.

(1) Changes to MEM mode.
(2) Execution of macro program registered in memory.
(3) Return to the mode before macro program is executed.
And the program which had been selected before macro program is executed is automatically selected.

This function is enabled only in the reset status (not during a reset). That is, this function cannot be used during automatic operation (including stopping and suspending of automatic operation), a reset, or an emergency stop.

Explanation

- Sequence between PMC and CNC

The signal must be processed between PMC and CNC according to the following procedures.

Start

(1) The macro call start signal MCSTx <Gn512.0 to Gn513.7> is input from PMC to CNC based on the signal from the macro call switch installed in the machine.

Mode change

(2) CNC outputs the mode notification signal and the mode change request signal MCRQ <Fn512.1> when the macro call start signal (MCSTx) is detected. At this time, MEM mode is notified as the mode notification signal. The macro call executing signal MCEXE <Fn512.0> and Call program confirmation signal MCEXx <Fn514.0 toFn515.7> are output at the same time.
(3) PMC must change the mode based on the signal output in the step of above (2).
(4) Set “1” in the mode change completion signal MCFIN <Gn514.0> by PMC when the mode change is completed. If this mode is different from that specified from the CNC in (2) or the reset mode, an alarm PS5306 occurs.

Execution of macro program

(5) When MCFIN signal is set to “1”, CNC executes the macro program.

End of macro program

(6) Instruct M02 or M30 at the end of the macro program.
Moreover, input external reset signal (ERS) or reset&rewind signal (RRW) with M02 or M30 on the PMC side.
Upon completion of the reset, the program selected before execution of the macro is automatically selected. The mode change request signal(MCRQ) and mode notification signals are output at the same time. At this time, the mode when macro call start signal (MCSTx) is input is notified as the mode notification signal.

Return of mode

(7) Change the mode on the PMC side based on the signal output in the step of above (6).
(8) Set “1” in Mode change completion signal (MCFIN) on the PMC side when the mode change is completed. On the CNC side, the Mode change request signal (MCRQ), the Macro call executing signal (MCEXE) and the Call program confirmation signal(MCEXx) are set to “0” concurrently with asserting of the Mode change completion signal (MCFIN). The PS5306 alarm is not checked at this time.

The Fig. 16.7 (a) shows the above-mentioned sequence.

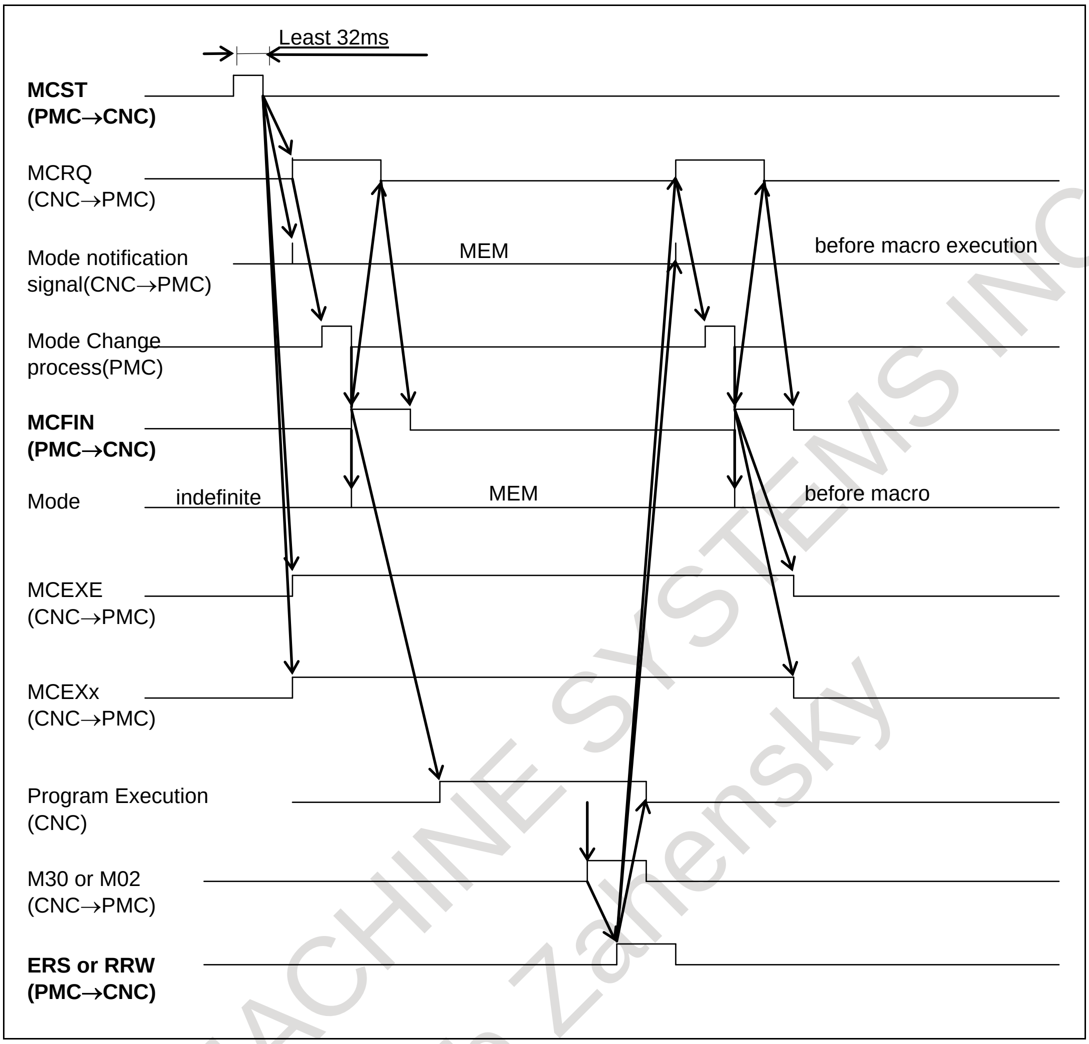


Fig. 16.7 (a)

- Interruption of sequence

Interruption by reset or emergency stop

When the started macro is interrupted by a reset or emergency stop before being completed, the abnormal end signal (MCSP) becomes “1” and stops the execution of the program. Then, when the reset is completed or the emergency stop is cleared, if the MEM mode is entered, the mode change request signal (MCRQ) and the mode notification signal are output for mode recovery and the selection program is recovered. (In a non-MEM mode, the system waits until the MEM mode is entered.) On the PMC side, be sure to set the mode change completion signal (MCFIN) to “1” to complete the sequence, regardless of whether mode recovery is made.

Feed hold or single block

The abnormal end signal (MCSP) is not output when stopping in feed hold or single block. Under such a condition, when the cycle start signal (ST) is turned on and off, the continuity of the macro program is executed.

Under such a condition, even if macro call start signal (MCSTx) is turned on and off, it is not effective.

Under such a condition, when reset or emergency stop are input, the operation which is described in “Interruption by reset or emergency stop” is executed.

Stop by alarm

When the execution of macro program is stopped by alarm, the abnormal end signal (MCSP) is output. Under such a condition, when reset or the emergency stop are input, the operation which is described in "Interruption by reset or emergency stop" is executed.

Fig. 16.7 (b) shows the timing chart of each signal when the sequence is interrupted by alarm.

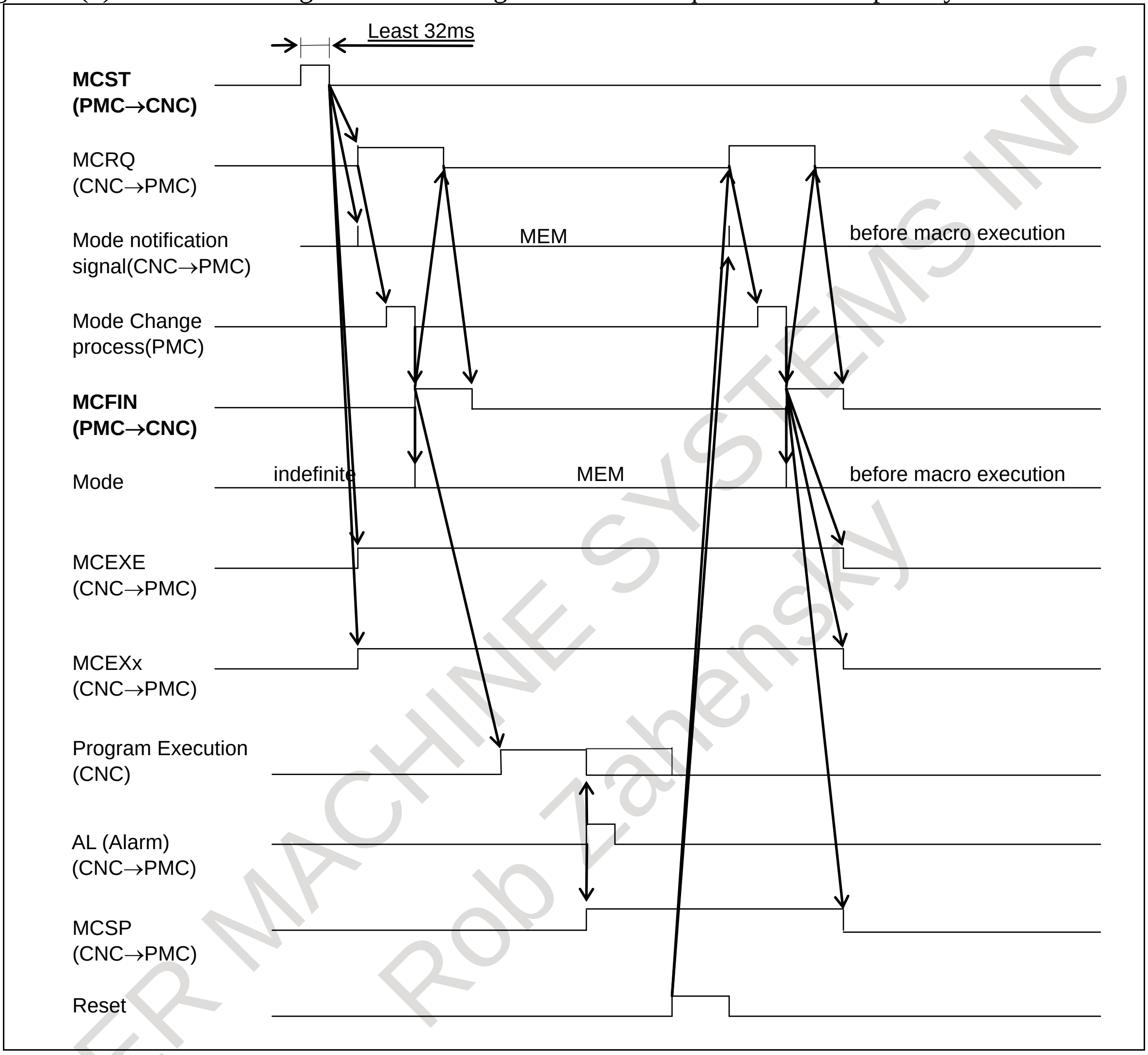


Fig. 16.7 (b)

Notes

NOTE

1. Even if the macro call is being executed, mode selection signal (MD1,MD2,MD4) is effective. Therefore, change the LADDER program to disable the mode change when the macro call executing signal (MCEXE) is "1" when the inconvenience is caused if the mode change is done.
2. The macro call start signal (MCSTx) is effective only the reset state. The macro call cannot be started during automatic operation (automatic operation stop state, automatic operation suspend state, and automatic operation start state), a reset, or an emergency stop. If an attempt is made to start macro program during automatic operation, a reset, or an emergency stop, an alarm PS5306 occurs.

NOTE

3 The called macro program must end with M02 or M30. And input external reset signal (ERS) or reset & rewind signal (RRW) from the PMC side when these M codes are executed. If above two condition is not implemented, neither the return of the mode nor the return of the selection program are done after the program ends.

4 When the specified program is not registered in the memory, the alarm PS0059 is generated. The operation in this case becomes the shown in “Stop by alarm”.

5 If a main program had been not selected when the one touch macro call function is started, the alarm DS0059 occurs at the end of execution of macro program. At this time, current folder becomes the folder where an one-touch macro exists, and the called one-touch macro remains being selected.

6 One Touch Macro programs need to be install to any folder among LIBRARY/, MTB2/, MTB1/, and SYSTEM/. When One Touch Macro program does not exist in any folder among LIBRARY/, MTB2/, MTB1/, and SYSTEM/, the alarm DS0059 "SPECIFIED NUMBER NOT FOUND" occurs at searching program. These folders to be searched are set in the bits 3 (SYS), bit 2 (MC1), bit 1 (MC2), and bit 0 (LIB) of parameter No. 3457 in advance. Even when a folder which a program is install to is LIBRARY/, MTB2/, MTB1/, or SYSTEM/, if the folder is not set as the search targets, the program is not searched for.

Signal

Macro call start signal MCSTx <Gn512.0 to Gn513.7>

[Classification] Input signal

[Function] This signal starts the macro call sequence.
When the fall of this signal is detected, CNC starts the corresponding macro program. O number of the program started by the MCST1 signal is specified by parameter No. 6096.

[Operation] The sequence is shown in “**Sequence between PMC and CNC**”.

Mode change completion signal MCFIN <Gn514.0>

[Classification] Input signal

[Function] This signal notifies the completion of the mode change to CNC.

[Operation] CNC begins the execution of the macro program or complete the sequence.

Macro call executing signal MCEXE <Fn512.0>

[Classification] Output signal

[Function] This signal notifies the macro call function is being executed.

[Output cond.] This signal is set to 1 in the following case:

- When CNC detects the fall of Macro call start signal (MCSTx).

This signal is set to “0” in the following case:

- When the sequence is completed with the mode change completion signal MCFIN set to “1” after macro execution is terminated or is interrupted by a reset, emergency stop, or alarm.

Mode change request signal MCRQ <Fn512.1>

[Classification] Output signal
[Function] This signal requests the change of the mode.
[Output cond.] This signal is set to “1” in the following case:

- When CNC detects the fall of Macro call start signal (MCSTx).
- When M30 or M02 is executed in the macro program, and external reset signal (ERS) or reset&rewind signal (RRW) is input.
- When the sequence is interrupted by reset or emergency stop.

This signal is set to “0” in the following case:

- When Mode change completion signal (MCFIN) becomes ”1”.

Mode notification signal MD1R,MD2R,MD4R,DNCIR,ZRNR<Fn513.0,Fn513.1,Fn513.2,Fn513.5,Fn513.7>

[Classification] Output signal
[Function] This signal notifies the mode which should be changed.
[Output cond.] This signal is output in the following case:

- When CNC detects the fall of Macro call start signal (MCSTx).
- When M30 or M02 is executed in the macro program, and external reset signal (ERS) or reset&rewind signal (RRW) is input.
- When the sequence is interrupted by reset or emergency stop.

Abnormal end signal MCSP <Fn512.2>

[Classification] Output signal
[Function] This signal notifies the sequence of the macro call is terminated abnormally.
[Output cond.] This signal is set to “1” in the following case:

- When the sequence is interrupted by reset or emergency stop or alarm.

This signal is set to “0” in the following case:

- When the mode change completion signal (MCFIN) becomes “1”, and the sequence is completed.

Call program confirmation signal MCEXx <Fn514.0 toFn515.7>

[Classification] Output signal
[Function] This signal notifies the program number called by the macro call.
The signal which corresponds to macro call start signal (MCSTx) is output.
[Output cond.] This signal is set to “1” in the following case:

- While executing the sequence.

This signal is set to “0” in the following case:

- When the sequence is completed.

Signal Address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn512	MCST8	MCST7	MCST6	MCST5	MCST4	MCST3	MCST2	MCST1
Gn513	MCST16	MCST15	MCST14	MCST13	MCST12	MCST11	MCST10	MCST9
Gn514								MCFIN

	#7	#6	#5	#4	#3	#2	#1	#0
Fn512						MCSP	MCRQ	MCEXE
Fn513	ZRNR		DNCIR			MD4R	MD2R	MD1R
Fn514	MCEX8	MCEX7	MCEX6	MCEX5	MCEX4	MCEX3	MCEX2	MCEX1
Fn515	MCEX16	MCEX15	MCEX14	MCEX13	MCEX12	MCEX11	MCEX10	MCEX9

Parameter

6095	The number of programs used by the one touch macro call function

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 16

Specify the number of programs used by the one touch macro call function.
For instance, when three is set, macro call start signal MCST1, MCST2, and MCST3 is valid.
When 0 is specified, this function is invalid.

6096	The first O number of the program used by the one touch macro call function

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

Specify the first O number of the program used by the one touch macro call function.
When 9000 is set, for example, the relationship between macro call start signal MSCTx and the program number of a program started by the signal is as follows:

MCST1 signal : Starts O9000 (when 1 or a greater value is set in parameter No. 6095).
MCST2 signal : Starts O9001 (when 2 or a greater value is set in parameter No. 6095).
MCST3 signal : Starts O9002 (when 3 or a greater value is set in parameter No. 6095).
: : :
MCST15 signal: Starts O9014 (when 15 or a greater value is set in parameter No. 6095).
MCST16 signal: Starts O9015 (when 16 is set in parameter No. 6095).

	#7	#6	#5	#4	#3	#2	#1	#0
3457					SYS	MC1	MC2	LIB

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> The parameters LIB, MC2, MC1, and SYS are used to set a search folder for the following subprogram/macro calls:
> - Subprogram call based on an M code
> - Subprogram call based on a particular address
> - Subprogram call based on a second auxiliary function code
> - Macro call based on a G code
> - Macro call based on an M code
> - Macro call based on a T code
> - One-touch macro call

#0 LIB The common program directory "//CNC_MEM/USER/LIBRARY/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

#1 MC2 MTB dedicated directory 2 "//CNC_MEM/MTB2/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

#2 MC1 MTB dedicated directory 1 "//CNC_MEM/MTB1/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

#3 SYS The system directory "//CNC_MEM/SYSTEM/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

Alarm and message

Number	Message	Description
DS0021	START ERROR(ONE TOUCH MACRO)	An One Touch Macro can not be start for the following reasons: (1) Feed hold signal *SP is0. (2) During alarm status. (3) Program restart signal SRN is "1". etc.
DS0059	SPECIFIED NUMBER NOT FOUND	The program of the specified O number is not registered in the memory.
PS5306	MODE CHANGE ERROR	The mode is not changed correctly at start of sequence. An one touch macro was started during a state other than the reset state, a reset, or an emergency stop.

16.8 PMC WINDOW PARAMETER WRITE

16.8.1 Parameter Write

Overview

You can write parameters by using PMC window function (function instruction WINDW: SUB52). You can also use FOCAS2 or C Language Executor to write parameters. However, all parameters cannot be written at any time. The following parameters cannot be written when any axes move.
Even if all axis have stopped, the following parameters cannot be written when beginning and ending automatic operation.
When the control axis is selected by the manual handle feed axis selection signals (HS1A - HS1D) in the manual handle feed mode, the axis is movement state and the following parameters cannot be written.
The error code is returned when the following parameter is written by PMC window function.
The error code EW_NUMBER(3) is returned when the following parameter is written by FOCAS2 or C Language Executor.
Refer to the parameter manual for the meaning of each parameter.
Examine it enough not to occur any trouble when the parameter is changed in operation even if any axes are not moving.

NOTE
The following list might be changed or added by adding new CNC function.

Table 16.8 (b) The list of parameters which cannot be written when any axes move

0	1	12	981	982	1000	1001	1002	1005	1006	1007
1008	1012	1013	1014	1015	1020	1022	1023	1025	1026	1031
1203	1205	1206	1220	1221	1222	1223	1224	1225	1226	1240
1241	1242	1243	1244	1250	1260	1280	1300	1301	1310	1320
1321	1322	1323	1324	1325	1326	1327	1350	1351	1352	1353
1354	1355	1356	1357	1358	1359	1360	1361	1401	1402	1403
1404	1405	1407	1408	1410	1414	1415	1420	1421	1423	1424
1425	1426	1427	1428	1430	1432	1434	1440	1441	1442	1443
1444	1445	1450	1451	1452	1453	1454	1455	1456	1457	1458
1459	1460	1461	1465	1466	1474	1481	1482	1483	1484	1485
1486	1487	1488	1495	1601	1602	1603	1604	1606	1610	1611
1612	1620	1621	1622	1623	1624	1625	1626	1627	1660	1671
1672	1673	1713	1714	1722	1726	1732	1735	1737	1738	1763
1769	1772	1783	1788	1789	1790	1791	1800	1802	1803	1804
1805	1814	1815	1816	1817	1818	1819	1820	1821	1822	1823
1825	1826	1827	1828	1829	1830	1832	1836	1838	1839	1841
1844	1846	1847	1848	1850	1851	1852	1868	1880	1881	1882
1883	1884	1885	1886	1902	1904	1905	2000	2002	2005	2007
2008	2009	2011	2015	2016	2017	2020	2021	2022	2023	2024
2025	2026	2027	2028	2029	2030	2031	2032	2033	2034	2035
2036	2037	2038	2039	2040	2041	2042	2043	2044	2045	2046
2047	2048	2049	2050	2051	2052	2053	2054	2055	2056	2057
2058	2059	2061	2062	2063	2064	2065	2066	2067	2068	2069
2070	2071	2072	2073	2074	2075	2076	2077	2078	2079	2080
2081	2082	2083	2084	2085	2086	2087	2088	2089	2090	2091
2092	2093	2094	2095	2096	2097	2098	2099	2100	2101	2102
2103	2104	2105	2106	2107	2108	2109	2110	2111	2112	2113
2114	2115	2116	2117	2118	2119	2120	2121	2122	2123	2124
2126	2127	2128	2129	2130	2131	2132	2133	2134	2135	2136
2137	2138	2139	2140	2142	2143	2144	2145	2146	2147	2148
2149	2150	2151	2152	2153	2154	2155	2156	2157	2158	2159
2160	2161	2162	2163	2164	2165	2166	2167	2168	2169	2170
2171	2172	2173	2174	2175	2176	2177	2178	2179	2180	2181
2182	2183	2184	2185	2186	2187	2188	2189	2190	2191	2192
2193	2194	2195	2196	2197	2198	2199	2206	2207	2224	2229
2230	2231	2232	2233	2234	2235	2236	2237	2238	2239	2240
2241	2242	2243	2244	2245	2246	2247	2248	2249	2250	2251
2252	2253	2254	2255	2256	2257	2258	2259	2260	2261	2262
2263	2264	2265	2266	2267	2268	2269	2270	2271	2273	2274
2276	2277	2278	2279	2280	2281	2282	2284	2285	2286	2287
2288	2289	2290	2291	2292	2293	2294	2295	2296	2297	2298
2299	2300	2302	2303	2304	2305	2306	2307	2308	2309	2310
2311	2312	2313	2314	2315	2316	2317	2318	2319	2320	2321
2322	2323	2324	2325	2326	2327	2328	2329	2330	2331	2332
2333	2334	2335	2336	2337	2338	2339	2340	2341	2342	2343
2344	2345	2346	2347	2348	2349	2350	2351	2352	2353	2354
2355	2356	2357	2358	2359	2360	2361	2362	2363	2364	2365
2366	2367	2368	2369	2370	2371	2372	2373	2374	2375	2376
2377	2378	2379	2380	2381	2382	2383	2384	2385	2386	2387
2388	2389	2390	2391	2392	2393	2394	2395	2396	2397	2398
2399	2400	2401	2402	2403	2404	2405	2406	2407	2408	2409
2410	2411	2412	2413	2414	2415	2416	2417	2418	2419	2420
2421	2422	2423	2424	2425	2426	2427	2428	2429	2430	2431
2432	2433	2434	2435	2436	2437	2438	2439	2440	2441	2442
2444	2445	2446	2447	2448	2449	2450	2451	2452	2453	2454

2455	2456	2457	2458	2459	2460	2461	2462	2463	2464	2465
2466	2467	2468	2469	2470	2471	2472	2473	2474	2475	2476
2477	2478	2479	2480	2481	2482	2483	2484	2485	2486	2487
2488	2489	2490	2491	2492	2493	2494	2495	2496	2497	2498
2499	2500	2501	2502	2503	2504	2505	2506	2507	2508	2509
2510	2511	2512	2513	2514	2515	2516	2517	2518	2519	2520
2521	2522	2523	2524	2525	2526	2527	2528	2529	2530	2531
2532	2533	2534	2535	2536	2537	2538	2539	2540	2541	2542
2543	2544	2545	2546	2547	2548	2549	2550	2551	2552	2553
2554	2555	2556	2557	2559	2560	2561	2562	2563	2564	2565
2566	2567	2568	2569	2570	2571	2572	2573	2574	2575	2576
2577	2578	2579	2580	2581	2582	2583	2584	2585	2586	2587
2588	2589	2590	2591	2592	2593	2594	2595	2596	2597	2598
2599	2600	2601	2602	2603	2604	2605	2606	2607	2608	2609
2610	2611	2612	2613	2614	2615	2616	2617	2618	2619	2620
2621	2622	2623	2624	2625	2626	2627	2628	2629	2630	2631
2632	2633	2634	2635	2636	2637	2638	2639	2640	2641	2642
2643	2644	2645	2646	2647	2648	2649	2650	2651	2652	2653
2654	2655	2656	2657	2658	2659	2660	2661	2662	2663	2664
2665	2666	2667	2668	2669	2670	2671	2672	2673	2674	2675
2676	2677	2678	2679	2680	2681	2682	2683	2684	2685	2686
2687	2688	2689	2690	2691	2692	2693	3006	3012	3013	3014
3019	3021	3022	3030	3031	3032	3033	3034	3035	3115	3129
3131	3132	3133	3134	3400	3401	3402	3405	3450	3452	3453
3455	3456	3457	3460	3463	3464	3471	3605	3620	3621	3622
3623	3624	3625	3626	3627	3661	3666	3671	3676	3681	3704
3715	3716	3717	3718	3719	3720	3721	3722	3730	3731	3738
3739	3741	3742	3743	3744	3772	3781	3794	3795	3796	3798
4352	4395	4801	4810	4811	4821	4826	4831	4832	4950	4959
4960	4961	4962	4963	4964	4970	4971	4972	4973	4974	5019
5022	5040	5041	5042	5043	5102	5148	5176	5177	5178	5179
5180	5181	5182	5183	5184	5185	5186	5187	5203	5210	5214
5221	5222	5223	5224	5231	5232	5233	5234	5241	5242	5243
5244	5261	5262	5263	5264	5271	5272	5273	5274	5280	5281
5282	5283	5284	5291	5292	5293	5294	5300	5301	5302	5304
5306	5310	5311	5312	5313	5321	5322	5323	5324	5350	5352
5354	5356	5358	5360	5365	5366	5367	5368	5382	5401	5420
5421	5431	5440	5460	5461	5463	5464	5480	5481	5484	5490
5500	5501	5510	5511	5512	5620	5761	5762	5763	5764	5771
5772	5773	5774	5781	5782	5783	5784	5861	5862	5863	5864
5871	5872	5873	5874	6007	6131	6132	6136	6137	6138	6161
6162	6163	6164	6165	6166	6167	6171	6172	6173	6174	6175
6176	6177	6181	6182	6183	6184	6185	6186	6187	6191	6192
6193	6194	6195	6196	6197	6201	6210	6280	6286	6287	6400
6402	6411	6412	6413	6414	6415	6416	6417	6418	6419	6420
6421	6422	6423	6424	6425	6426	6427	6428	6429	6430	6431
6432	6433	6434	6435	6436	6437	6438	6439	6440	6441	6442
6443	6444	6445	6446	6447	6448	6449	6450	6451	6452	6453
6454	6455	6456	6457	6458	6459	6460	6461	6462	6463	6464
6465	6466	6467	6468	6469	6470	6471	6472	6473	6474	6475
6476	6477	6478	6479	6480	6481	6482	6483	6484	6485	6486
6487	6488	6489	6490	6495	6496	6497	7015	7040	7041	7042
7055	7066	7102	7106	7160	7161	7180	7181	7182	7183	7184
7185	7186	7187	7188	7301	7310	7501	7502	7503	7504	7505
7510	7514	7515	7521	7522	7523	7524	7525	7527	7570	7605
7635	7636	7700	7702	7704	7705	7709	7710	7731	7740	7741

7745	7772	7773	7776	7777	7778	7782	7783	7784	7785	8002
8003	8004	8005	8006	8007	8008	8010	8011	8013	8020	8022
8028	8030	8031	8032	8040	8101	8105	8161	8162	8163	8164
8166	8167	8168	8169	8180	8181	8183	8184	8185	8186	8190
8191	8192	8194	8210	8211	8212	8303	8304	8305	8307	8311
8312	8314	8323	8325	8326	8327	8330	8331	8332	8333	8334
8335	8336	8375	8377	8406	8407	8451	8456	8457	8458	8459
8465	8466	8486	8487	8490	8491	8860	8861	10000	10001	10002
10003	10004	10005	10006	10007	10008	10009	10010	10011	10012	10013
10014	10015	10016	10017	10018	10019	10020	10021	10022	10023	10024
10025	10026	10027	10028	10029	10030	10031	10032	10033	10034	10035
10036	10037	10038	10039	10040	10041	10042	10043	10044	10045	10046
10047	10048	10049	10050	10051	10052	10053	10054	10055	10056	10057
10058	10059	10060	10061	10062	10063	10064	10065	10066	10067	10068
10069	10070	10071	10072	10073	10074	10075	10076	10077	10078	10079
10080	10081	10082	10083	10084	10085	10086	10087	10088	10089	10090
10091	10092	10093	10094	10095	10096	10097	10098	10099	10100	10101
10102	10103	10104	10105	10106	10107	10108	10109	10110	10111	10112
10113	10114	10115	10116	10117	10118	10119	10120	10121	10122	10123
10124	10125	10126	10127	10128	10129	10130	10131	10132	10133	10134
10135	10136	10137	10138	10139	10140	10141	10142	10143	10144	10145
10146	10147	10148	10149	10150	10151	10152	10153	10154	10155	10156
10157	10158	10159	10160	10161	10162	10163	10164	10165	10166	10167
10168	10169	10170	10171	10172	10173	10174	10175	10176	10177	10178
10179	10180	10181	10182	10183	10184	10185	10186	10187	10188	10189
10190	10191	10192	10193	10194	10195	10196	10197	10198	10199	10200
10201	10202	10203	10204	10205	10206	10207	10208	10209	10210	10211
10212	10213	10214	10215	10216	10217	10218	10219	10220	10221	10222
10223	10224	10225	10226	10227	10228	10229	10230	10231	10232	10233
10234	10235	10236	10237	10238	10239	10240	10241	10242	10243	10244
10245	10246	10247	10248	10249	10250	10251	10252	10253	10254	10255
10256	10257	10258	10259	10260	10261	10262	10263	10264	10265	10266
10267	10268	10269	10270	10271	10272	10273	10274	10275	10276	10277
10278	10279	10280	10281	10282	10283	10284	10285	10286	10287	10288
10289	10290	10291	10292	10293	10294	10295	10296	10297	10298	10299
10300	10301	10302	10303	10304	10305	10306	10307	10308	10309	10310
10311	10312	10313	10314	10315	10316	10317	10318	10319	10320	10321
10322	10323	10324	10325	10326	10327	10328	10329	10331	10332	10339
10341	10342	10345	10346	10350	10360	10361	10362	10370	10371	10410
10411	10412	10480	10485	10486	10487	10490	10491	10500	10501	10502
10503	10504	10505	10506	10507	10508	10509	10510	10511	10512	10513
10514	10515	10516	10517	10518	10519	10520	10521	10522	10523	10524
10525	10526	10527	10528	10529	10530	10531	10532	10533	10534	10535
10536	10537	10538	10539	10540	10541	10542	10543	10544	10545	10546
10547	10548	10549	10550	10551	10552	10553	10554	10555	10556	10557
10558	10559	10560	10561	10562	10563	10564	10565	10566	10567	10568
10569	10570	10571	10572	10573	10574	10575	10576	10577	10578	10579
10580	10581	10582	10583	10584	10585	10586	10587	10588	10589	10590
10591	10592	10593	10594	10595	10596	10770	10771	10772	10773	10774
10775	10776	10777	10778	10779	10780	10781	10782	10783	10784	10790
10791	10792	10793	10794	10795	10796	10797	10800	10801	10802	10803
10804	10805	10806	10807	10808	10809	10810	10811	10812	10813	10814
10830	10900	10901	10902	10903	10904	10905	10906	10907	10908	10909
10910	10911	10912	10913	10914	10915	10916	10917	10918	10919	10920
10921	10926	10927	10928	11000	11001	11005	11009	11012	11013	11014
11015	11016	11017	11020	11021	11022	11023	11024	11025	11026	11030

11031	11032	11033	11034	11035	11036	11037	11040	11041	11042	11050
11051	11052	11060	11061	11062	11063	11065	11066	11067	11068	11070
11071	11072	11073	11074	11075	11076	11080	11081	11082	11083	11084
11085	11086	11087	11090	11101	11102	11113	11118	11119	11201	11202
11203	11221	11222	11230	11231	11232	11240	11242	11244	11245	11250
11260	11261	11263	11270	11275	11276	11280	11281	11282	11283	11284
11400	11401	11402	11403	11420	11421	11422	11423	11424	11425	11426
11427	11428	11429	11430	11431	11432	11433	11434	11435	11436	11437
11438	11439	11440	11441	11442	11443	11444	11445	11446	11448	11449
11450	11451	11452	11453	11454	11455	11456	11457	11458	11459	11460
11461	11462	11463	11464	11465	11466	11467	11468	11469	11470	11471
11472	11473	11474	11476	11477	11478	11479	11480	11485	11487	11488
11489	11490	11500	11501	11502	11504	11549	11601	11602	11619	11630
11768	11769	11770	11771	11772	11773	11774	11775	11776	11777	11778
11779	11780	11803	11807	12255	12256	12318	12319	12321	12600	12730
12731	12732	12733	12734	12735	12736	12737	12738	12751	12752	12753
12754	12755	12756	13113	13351	13352	13353	13354	13361	13362	13363
13364	13371	13372	13373	13374	13391	13392	13393	13394	13395	13396
13421	13541	13542	13600	13634	13681	13806	13811	13821	13822	13823
13824	13826	13827	13828	13829	13831	13832	13833	13834	13835	13836
13837	13838	13844	13880	13881	13882	13883	13884	13885	13886	13887
13888	13889	13890	13891	13892	13893	13894	13895	13896	13897	13898
13899	13900	13901	13902	13903	13904	13905	13906	13907	13908	13909
13910	13911	13920	13921	13922	13923	13924	13925	13926	13927	13928
13929	13930	13931	13932	13933	13934	13935	13936	13937	13938	13939
13940	13941	13942	13943	13944	13945	13946	13947	13948	13949	13950
13951	13960	13961	13962	13963	13964	13965	13966	13967	13968	13969
13970	13971	13972	13973	13974	13975	13976	13977	13978	13979	13980
13981	13982	13983	13984	13985	13986	13987	13988	13989	13990	13991
14010	14270	14271	14273	14274	14275	14276	14277	14278	14279	14280
14281	14282	14283	14284	14285	14286	14287	14288	14289	14310	14901
14902	14903	14910	14911	14912	14913	14914	14915	14916	14917	14918
14920	14921	14922	14923	14924	14925	14926	14927	14928	14929	14930
14931	14932	14933	14934	14935	14936	14937	14938	14940	14941	14942
14943	14944	14945	14946	14947	14948	14949	14950	14951	14952	14953
14954	14955	14956	14957	14958	14960	14961	14962	14963	14964	14965
14966	14967	14968	14969	14970	14971	14972	14973	14974	14975	14976
14977	14978	14985	14986	14987	14988	14997	14998	18500	18501	18502
18503	18505	18506	18507	18508	18509	18510	18511	18512	18513	18514
18515	18516	18517	18518	18519	18520	18521	18522	18523	18524	18525
18526	18527	18528	18529	18530	18531	18532	18533	18534	18535	18536
18537	18538	18539	18540	18545	18546	18547	18548	18550	18551	18552
18553	18554	18555	18556	18557	18558	18559	18560	18561	18562	18563
18564	18565	18566	18567	18568	18569	18570	18571	18572	18573	18574
18575	18576	18577	18578	18579	18580	18581	18582	18583	18584	18585
18590	18591	18592	18593	18595	18596	18597	18598	18599	18600	18601
18602	18603	18604	18605	18606	18607	18608	18609	18610	18611	18612
18613	18614	18615	18616	18617	18618	18619	18620	18621	18622	18623
18624	18625	18626	18627	18628	18629	18630	18640	18641	18642	18643
18645	18646	18647	18648	18650	18651	18652	18653	18655	18656	18657
18658	18659	18660	18661	18662	18663	18664	18665	18666	18667	18668
18669	18670	18671	18672	18673	18674	18675	18676	18677	18678	18679
18680	18681	18682	18683	18684	18685	18686	18687	18688	18689	18690
18691	18692	18693	18694	18695	18696	18697	18698	18699	18700	18701
18702	18703	18704	18705	18706	18707	18708	18709	18710	18711	18712
18713	18714	18715	18716	18717	18718	18719	18720	18721	18722	18723

18724	18725	18726	19500	19501	19503	19515	19516	19517	19518	19519
19531	19532	19533	19534	19535	19540	19541	19542	19543	19544	19545
19546	19547	19548	19549	19550	19551	19552	19553	19554	19555	19556
19557	19558	19559	19560	19561	19562	19563	19564	19565	19566	19567
19568	19581	19582	19587	19588	19589	19590	19591	19592	19593	19608
19650	19655	19656	19657	19658	19659	19660	19661	19662	19665	19666
19667	19680	19681	19682	19683	19684	19685	19686	19687	19688	19689
19690	19696	19697	19698	19699	19700	19701	19702	19703	19704	19705
19709	19710	19711	19712	19713	19714	19751	19752	19760	19761	19762
24096	24097	24098	24099	24100	24101	24102	24103	24604	24605	24607
24610	24611	24612	24613	24614	24615	24616	24617	24618	24619	24620
24621	24632	24633	24640	24653	24660	24711	24713	25700	25701	25702
25703	25704	25705	25706	25710	25711	25712	25713	25714	25715	25716
25717	25720	25721	25722	25723	25724	25725	25726	25730	25731	25732
25733	25734	25735	25736	25737	25861	25862	25863	25864	25865	25866
25867	25868	25869	25870	25884	25885	25886	25887			

16.8.2 Parameter (No. 2092, Bit 0 of No. 8162) Write

Overview

This function allows you to write the following parameters using a PMC window function (function instruction WINDW: SUB52) as long as a relevant axis is stopped. You can also use FOCAS2 or C Language Executor to write parameters. The function can be used to write the following parameters:

- Parameter No. 2092
  Advanced preview feed forward coefficient

- Bit 0 (SMR) of parameter No. 8162
  Synchronous mirror-image control is:
  0 : Not applied. (The master and slave axes move in the same direction.)
  1 : Applied. (The master and slave axes move in opposite directions.)

This function can reduce the machining cycle time as it can write parameters without all of the axes stopped.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11502	IPW							

[Input type] Parameter input
[Data type] Bit

#7 IPW The advanced preview feed-forward coefficient (parameter No. 2092) and bit 0 (SMR) of parameter No. 8162 for specifying whether to apply a mirror image during synchronization control are:
0: Write-disabled during axis moving.
1: Write-enabled if the corresponding axis is stopped.

Limitation

- Axis synchronous control

Under axis synchronous control, this function is disabled when slave axis parameter automatic setting is enabled (bit 4 (SYP) of parameter No. 8303 = 1). (All the axes must be stopped before parameters can be written.)

- AI contour control

Under AI contour control, no parameter can be written even when the relevant axis is stopped.

16.8.3 Parameter (No. 1620) Write

Overview

This function allows you to write a time constant (parameter No. 1620) for rapid-traverse linear acceleration/deceleration during axial movement. You can write the parameter using the PMC window function (function instruction WINDW: SUB52), FOCAS2, or C Language Executor.

If you change the time constant during axial movement, the new setting take effect from the move command encountered first after the axis stops. This function can reduce the machining cycle time as it can change the time constant without all of the axes stopped.

Example)

G00 X50.0 ;

G00 X150.0 ;

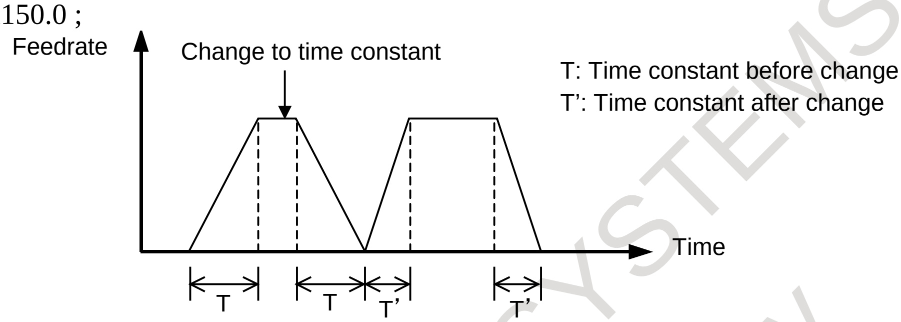


Fig. 16.8.3 (a)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11502		CTC						

[Input type] Parameter input
[Data type] Bit

#6 CTC During axis moving, the time constant of rapid traverse linear acceleration/deceleration for each axis (parameter No. 1620) is:

0: Write-disabled.
1: Write-enabled.

Restrictions

- Axis of time constant 0

Do not use this function for an axis of time constant 0.

- Axis synchronous control

Under axis synchronous control, this function is disabled when slave axis parameter automatic setting is enabled (bit 4 (SYP) of parameter No. 8303 = 1). (All the axes must be stopped before the parameter can be written.)

Note

NOTE

1 Set the time constants of the master and slave axes under axis synchronous control to the same value.
2 Set the time constants of the master and slave axes under tandem control to the same value.

16.8.4 Parameter (No. 1825, No. 1826) Write

Overview

This function allows you to write the following parameters using a PMC window function (function instruction WINDW: SUB52) as long as a relevant axis is stopped. You can also use FOCAS2 or C Language Executor to write parameters. The function can be used to write the following parameters:

- Parameter No. 1825
  Servo loop gain for each axis

- Parameter No. 1826
  In-position width for each axis

This function can reduce the machining cycle time as it can write parameters without all of the axes stopped.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
10350							PWR	

[Input type] Parameter input
[Data type] Bit

#1 IPW The servo loop gain for each axis (parameter No. 1825) and the In-position width for each axis (parameter No. 1826) are:
0: Write-disabled during axis moving.
1: Write-enabled if the corresponding axis is stopped.

> **NOTE**
> This parameter is Automatic setting parameter. For details, refer to the appendix "Automatic setting / Initial setting parameter" in this manual.

Limitation

- **Axis synchronous control**

Under axis synchronous control, this function is disabled when slave axis parameter automatic setting is enabled (bit 4 (SYP) of parameter No. 8303 = 1). (All the axes must be stopped before parameters can be written.)

16.9 DATA TRANSFER BETWEEN PMC AND DCSPMC

Overview

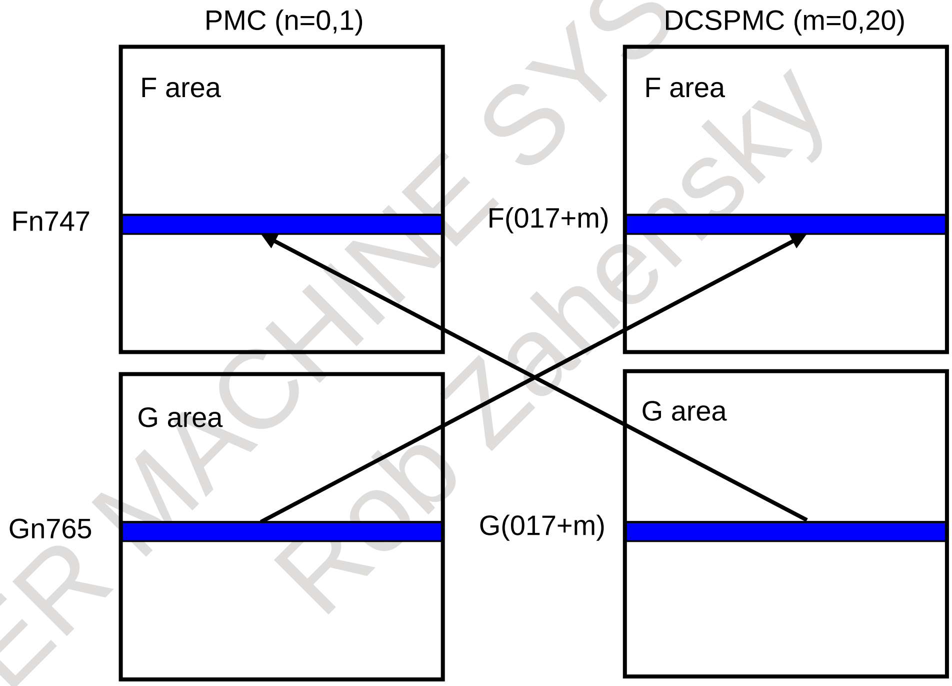
The signal data are transferred between PMC and DCSPMC. As a result, PMC data can be referred from DCSPMC and DCSPMC data can be referred from PMC.
Regardless of 1-path or the multi path system, 2 bytes (16 points) data are transferred.

Explanation

The signal data are transferred between PMC and DCSPMC as follows.

Data transfer area

PMC	G0765	→	DCSPMC	F0017	(1byte)
	G1765	→		F0037	(1byte)
DCSPMC	G0017	→	PMC	F0747	(1byte)
	G0037	→		F1747	(1byte)



> **NOTE**
> Regardless of 1-path or the multi path system, 2 bytes (16 points) are transferred.

Signal

DI signal for Data transfer between PMC and DCSPMC
TPMG00 to TPMG07 <PMC:Gn765> (n=0,1)
TDCG00 to TDCG07 <DCSPMC:G(017+m)> (m=0,20)

[Classification] Input signal

[Function] These signals are transferred between PMC and DCSPMC. PMC signals can be referred from DCSPMC and DCSPMC signals can be referred from PMC. The signals are transferred as follows.
- From TPMGxx(G765) to TPMFxx(F017).
- From TDCGxx(G017) to TDCFxx(F747).

[Operation] (1) DI signal(TPMG00 to TPMG07) are transferred to DO signal(TPMF00 to TPMF07)
(2) DI signal(TDCG00 to TDCG07) are transferred to DO signal(TDCF00 to TDCF07)

DO signal for Data transfer between PMC and DCSPMC
TDCF00 to TDCF07 <PMC:Fn747> (n=0,1)
TPMF00 to TPMF07 <DCSPMC:F(017+m)> (m=0,20)

[Classification] Output signal

[Function] These signals are transferred between PMC and DCSPMC. PMC signals can be referred from DCSPMC and DCSPMC signals can be referred from PMC. The signals are transferred as follows.
- From TPMGxx(G765) to TPMFxx(F017).
- From TDCGxx(G017) to TDCFxx(F747).

[Operation] (1) DI signal(TPMG00 to TPMG07) are transferred to DO signal(TPMF00 to TPMF07)
(2) DI signal(TDCG00 to TDCG07) are transferred to DO signal(TDCF00 to TDCF07)

Signal address

PMC (n=0,1)

	#7	#6	#5	#4	#3	#2	#1	#0
Gn765	TPMG07	TPMG06	TPMG05	TPMG04	TPMG03	TPMG02	TPMG01	TPMG00
Fn747	TDCF07	TDCF06	TDCF05	TDCF04	TDCF03	TDCF02	TDCF01	TDCF00

DCSPMC (m=0,20)

	#7	#6	#5	#4	#3	#2	#1	#0
G(017+m)	TDCG07	TDCG06	TDCG05	TDCG04	TDCG03	TDCG02	TDCG01	TDCG00
F(017+m)	TPMF07	TPMF06	TPMF05	TPMF04	TPMF03	TPMF02	TPMF01	TPMF00

⚠ **WARNING**

1 The safety related signal such as the protection door lock/unlock signal is input(or output) to PMC/DCSPMC respectively and monitored in redundant mode. If this transfer function is applied to the safety related signal, the independence between PMC/DCSPMC is lost. Do not use this data transfer function to control the safety related function.

2 The error of ladder program cannot be checked by safety function itself. The confirmation of the safety related ladder program should be done enough by the machine tool builder. The safety related ladder program should be guaranteed by the machine tool builder.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
13805		PDTR	NPDT					

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#5 NPDT While the Dual Check Safety function is inactive (parameter DCE=0), Data transfer between PMC and DCSPMC is:
0: Not available
1: Available

#6 PDTR Data transfer between PMC and DCSPMC is:
0: Not available
1: Available

16.10 KEY HIT STATUS SIGNAL OUTPUT FUNCTION

Overview

This function outputs the status of key hit to PMC signal when the soft key and MDI key are hit.

> ⚠ **CAUTION**
> 1 The delay from hitting key to outputting of PMC signal is 100ms or less. Therefore, this function can not be used to operation that is required to quick key response.
> 2 Only when a CNC hardware key is pushed, the PMC signal is output.
> 3 When CNC hardware key is pushed while using CNC screen display function with PC connected by HSSB or Ethernet, the PMC signal is output.
> 4 On Stand-alone type CNC (Windows CE), when display and CNC are connected by Ethernet, the PMC signal is not output.
> 5 This function is not available on NCGuide / NCGuidePro.

Explanation

Key hit status is output to PMC signal when soft key and MDI key are hit.

- Key which outputs the hit status

Key which outputs hit status by this function is as follows.

Kind of key	Key	PMC signal (Symbol)
Digit key	0	KEY_DG0
	1	KEY_DG1
	2	KEY_DG2
	3	KEY_DG3
	4	KEY_DG4
	5	KEY_DG5
	6	KEY_DG6
	7	KEY_DG7
	8	KEY_DG8
	9	KEY_DG9

Kind of key	Key	PMC signal (Symbol)
Horizontal soft key (See Fig. 17.11)	HSKY1	KEY_HS1
	HSKY2	KEY_HS2
	HSKY3	KEY_HS3
	HSKY4	KEY_HS4
	HSKY5	KEY_HS5
	HSKY6	KEY_HS6
	HSKY7	KEY_HS7
	HSKY8	KEY_HS8
	HSKY9	KEY_HS9
	HSKY10	KEY_HS10
	HSKYL	KEY_HSL
	HSKYR	KEY_HSR
Vertical soft key (See Fig. 17.11)	VSKY1	KEY_VS1
	VSKY2	KEY_VS2
	VSKY3	KEY_VS3
	VSKY4	KEY_VS4
	VSKY5	KEY_VS5
	VSKY6	KEY_VS6
	VSKY7	KEY_VS7
	VSKY8	KEY_VS8
	VSKY9	KEY_VS9
Cursor key	→	KEY_CRR
	←	KEY_CRL
	↓	KEY_CRD
	↑	KEY_CRU
Page key	⇊	KEY_PGD
	⇈	KEY_PGU
Other key	SHIFT	KEY_SFT

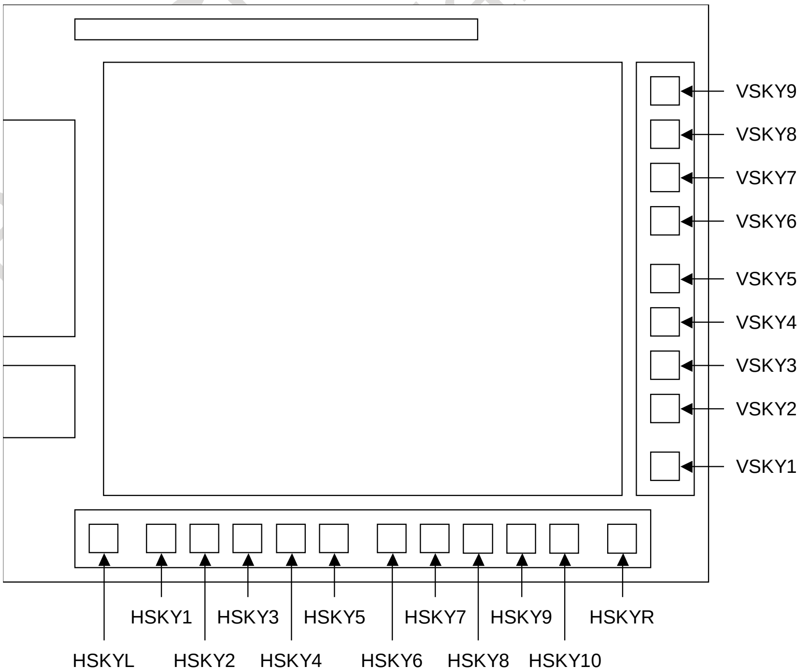


Fig. 16.10 (a) Arrangement of soft key (10.4” display unit, 15” display unit)

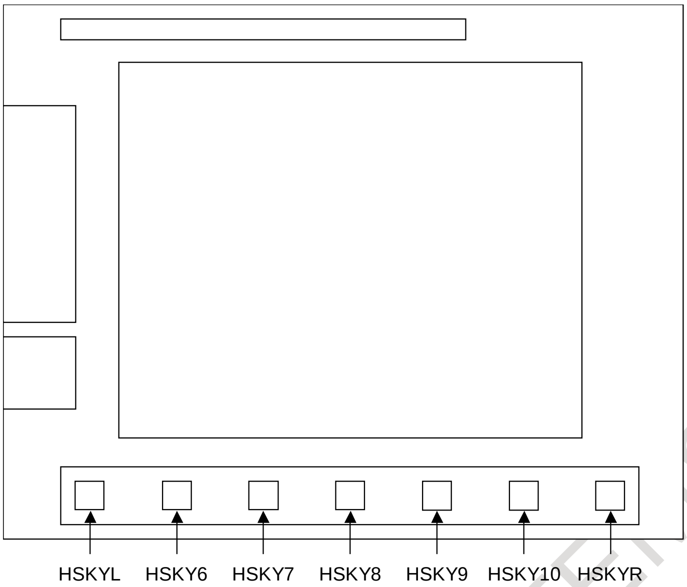


Fig. 16.10 (b) Arrangement of soft key (8.4” display unit)

> **NOTE**
> Hit status is not output to PMC signal if the key which is not supported is hit.

- Output format of PMC signal

The key hit status is output to PMC signal which address kind is set by parameter No.13125 and which address offset is set by parameter No.13126 as follows.

	#7	#6	#5	#4	#3	#2	#1	#0
+0	KEY_DG7	KEY_DG6	KEY_DG5	KEY_DG4	KEY_DG3	KEY_DG2	KEY_DG1	KEY_DG0
+1							KEY_DG9	KEY_DG8
+2	KEY_HS8	KEY_HS7	KEY_HS6	KEY_HS5	KEY_HS4	KEY_HS3	KEY_HS2	KEY_HS1
+3					KEY_HSL	KEY_HSR	KEY_HS10	KEY_HS9
+4	KEY_VS1	KEY_VS2	KEY_VS3	KEY_VS4	KEY_VS5	KEY_VS6	KEY_VS7	KEY_VS8
+5								KEY_VS9
+6	KEY_SFT		KEY_PGU	KEY_PGD	KEY_CRU	KEY_CRD	KEY_CRL	KEY_CRR
+7								

> **NOTE**
> 1 Blank bit in above table is not used.
> 2 Key hit status is not output to PMC signal when 3 or more keys are hit in same time.

- Supported hardware

This function is available on the following hardware.

- Soft key

  Hit status of the soft key is output to PMC signal if the following display unit are used.

CNC type	Kind of display unit
LCD mounted type CNC	8.4” color LCD (without Touch panel) 10.4” color LCD A (without Touch panel) 10.4” color LCD A (with Touch panel) 10.4” color LCD B (without Touch panel) 10.4” color LCD B (with Touch panel) 15” color LCD (without Touch panel) 15” color LCD (with Touch panel)

CNC type	Kind of display unit
LCD mounted type CNC (Windows CE function)	10.4” color LCD (without Touch panel) 10.4” color LCD (with Touch panel) 12.1” color LCD (without Touch panel) 12.1” color LCD (with Touch panel) 15” color LCD (without Touch panel) 15” color LCD (with Touch panel)
Stand-alone type CNC	10.4” color LCD A (without Touch panel) 10.4” color LCD A (with Touch panel) 10.4” color LCD B (without Touch panel) 10.4” color LCD B (with Touch panel) 15” color LCD (without Touch panel) 15” color LCD (with Touch panel)
Stand-alone type CNC (Windows CE function)	Display Unit with WindowsCE (10.4” LCD without Touch panel) Display Unit with WindowsCE (10.4” LCD with Touch panel) Display Unit with WindowsCE (12.1” LCD without Touch panel) Display Unit with WindowsCE (12.1” LCD with Touch panel) Display Unit with WindowsCE (15” LCD without Touch panel) Display Unit with WindowsCE (15” LCD with Touch panel)

NOTE

The hit status of soft key is not output to PMC signal in the following case.

- Soft key on touch panel is hit.
- The display unit that is not supported is used.
- On Stand-alone type CNC (Windows CE), the display and the CNC are connected by Ethernet.

- MDI key

  Hit status of the MDI key is output to PMC signal if the following MDI unit is used.

CNC type	Kind of MDI unit
LCD mounted type CNC	MDI unit (Turning, ONG, vertical) MDI unit (Milling, ONG, vertical) MDI unit (QWERTY type A, vertical) MDI unit (QWERTY type B, vertical) MDI unit (Turning, ONG, horizontal) MDI unit (Milling, ONG, horizontal) MDI unit (Turning, ONG, vertical/horizontal) MDI unit (Milling, ONG, vertical/horizontal) MDI unit (Turning, small type, horizontal) MDI unit (Milling, small type, horizontal)
LCD mounted type CNC (Windows CE function)	MDI unit (Turning, ONG, vertical) MDI unit (Milling, ONG, vertical) MDI unit (QWERTY type A, vertical) MDI unit (QWERTY type B, vertical) MDI unit (Turning, ONG, horizontal) MDI unit (Milling, ONG, horizontal) MDI unit (Turning, ONG, vertical/horizontal) MDI unit (Milling, ONG, vertical/horizontal)
Stand-alone type CNC	MDI unit (Turning, ONG, vertical) MDI unit (Milling, ONG, vertical) MDI unit (QWERTY type A, vertical) MDI unit (QWERTY type B, vertical) MDI unit (Turning, ONG, horizontal) MDI unit (Milling, ONG, horizontal)

CNC type	Kind of MDI unit
Stand-alone type CNC (Windows CE function)	MDI unit (Turning, ONG, vertical) MDI unit (Milling, ONG, vertical) MDI unit (QWERTY type A, vertical) MDI unit (QWERTY type B, vertical) MDI unit (Turning, ONG, horizontal) MDI unit (Milling, ONG, horizontal)

NOTE

The hit status of MDI key is not output to PMC signal in the following case.
- Key was hit on touch panel.
- The MDI unit or keyboard that is not supported is used.
- The MDI unit which is connected via PANEL *i* is used.

Parameter

13125	Address type of PMC signal area which outputs the key hit status

[Input type] Parameter input
[Data type] Byte
[Valid data range] 1、2、11、21、31、41

Set the address type of PMC signal area which outputs the key hit status.
The relation between the setting value and the address type is as follows.

Setting value	Address type
1	R address in 1st PMC
2	E address in 1st to 5th PMC
11	R address in 2nd PMC
21	R address in 3rd PMC
31	R address in 4th PMC
41	R address in 5th PMC

NOTE

1 The key hit status is not output if the setting value is out of range.
2 When this parameter is set, the power must be turned off before operation is continued.

13126	Top address of PMC signal area which outputs the key hit status

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to Maximum address

Set the Top address of PMC signal area which outputs the key hit status.
A continuous area in 8 bytes is necessary to output the key hit status.

The maximum address in the PMC address is as follows.

	1st-5th PMC			
	PMC memory A	PMC memory B	PMC memory C	PMC memory D
R address	R0 to R1499	R0 to R7999	R0 to R15999	R0 to R59999
E address	E0 to E9999	E0 to E9999	E0 to E9999	E0 to E9999

> **NOTE**
> 1 The key hit status is not output if the setting value is out of range.
> 2 When this parameter is set, the power must be turned off before operation is continued.

16.11 SCREEN NUMBER NOTIFICATION FUNCTION

Overview

This function outputs the screen number (Large classification number and Small classification number) of the screen that is currently displayed to PMC signal.

Signal

Screen large classification notification signal SCRNL0 to SCRNL7<F523>

[Classification] Output signal
[Function] Large classification of screen number is notified.
[Operation] The following screen numbers (Large classification number) can be read.

Large classification	Screen	Function key
0	Position screen	POSITION
1	Program screen	PROGRAM
2	Offset/setting screen	OFFSET
3	System screen	SYSTEM
4	Message screen	MESSAGE
5	Graphics screen	GRAPHIC
6	Custom screen 1	CUSTOM1
7	Custom screen 2	CUSTOM2
8	Main menu	—

Screen small classification notification signal SCRNS0 to SCRNS7<F524>

[Classification] Output signal
[Function] Small classification of screen number is notified.
[Operation] The following screen numbers (Small classification number) can be read.

Large classification	Small classification	Content
0	01H	Current position (absolute coordinate) screen
	02H	Current position (relative coordinate) screen
	03H	Current position (overall) screen
	04H	Handle interruption screen
	06H	Operating monitor screen
	07H	3-dimensional Manual Feed Screen
	0BH	Current position (absolute coordinate) screen (15inch,19inch)
	0CH	Current position (relative coordinate) screen (15inch,19inch)
	0DH	Smart adaptive control screen
	32H	C Language Executor application screen 1
	33H	C Language Executor application screen 2
	34H	C Language Executor application screen 3
	35H	C Language Executor application screen 4
	36H	C Language Executor application screen 5

Large classification	Small classification	Content
1	01H	Screen displaying a program
	02H	Program folder screen
	03H	Next block display screen
	04H	Program check screen
	06H	Machining time display screen
	08H	Program restart screen
	09H	Robot connection Robot program select screen
	0AH	Current block display screen
	32H	C Language Executor application screen 1
	33H	C Language Executor application screen 2
	34H	C Language Executor application screen 3
	35H	C Language Executor application screen 4
	36H	C Language Executor application screen 5
2	01H	Tool offset screen
	02H	Setting screen
	03H	Work coordinates screen
	06H	Custom macro screen
	07H	Pattern menu screen
	08H	Software operator's panel screen
	09H	Displaying tool life management (list screen)
	0BH	Y-axis offset screen
	0CH	Workpiece coordinate system shift screen
	0DH	Tool offset/second geometry tool offset screen
	0EH	Tool geometry data screen
	11H	Precision level selection screen/
	12H	4th/5th axis offset screen
	15H	Chuck barrier setting screen
	16H	Language screen
	17H	Protection of data at 8levels screen
	18H	Operation confirmation function setting screen
	1DH	Tool geometry size data setting screen
	1FH	Workpiece setting error screen
	24H	Dynamic tool offset screen
	27H	Machine configuration selecting screen
	28H	Machining condition selection screen
	29H	Eco setting screen
	2CH	Machining setting switched screen
	32H	C Language Executor application screen 1
	33H	C Language Executor application screen 2
	34H	C Language Executor application screen 3
	35H	C Language Executor application screen 4
	36H	C Language Executor application screen 5

Large classification	Small classification	Content
3	01H	Parameter screen
	02H	Diagnosis screen
	03H	Servo guide mate screen
	04H	System configuration screen
	06H	Memory contents display screen
	07H	Pitch error compensation screen
	08H	Servo parameter setting screen
	09H	Spindle setting screen
	0BH	PMC diagnosis/maintenance screen
	0CH	Ladder diagram monitor and editor screens
	0DH	PMC configuration data setting screens
	10H	Machining parameter setting screen
	11H	ALL IO (RS-232C) screen
	12H	ALL IO screen
	13H	Operation history screen
	15H	Color setting screen
	16H	Periodic maintenance screen
	17H	Maintenance information screen
	18H	Waveform Diagnosis Screen
	1AH	Touch panel calibration screen
	1BH	FSSB setting screen
	1CH	Menu screen for parameter tuning
	1DH	Power Mate CNC manager screen
	1FH	Embedded Ethernet port screen
	20H	PCMCIA Ethernet card screen
	21H	Ethernet screen
	22H	PROFIBUS-DP Master screen
	24H	Machine Remote Diagnosis screen
	25H	M code group setting screen
	29H	PROFIBUS-DP Slave screen
	2AH	DeviceNet Master screen
	2BH	FL-net (1st) screen
	2CH	DeviceNet Slave screen
	2EH	Dual Check Safety screen
	2FH	Real time macro screen
	30H	High pitch error compensation screen
	32H	C Language Executor application screen 1
	33H	C Language Executor application screen 2
	34H	C Language Executor application screen 3
	35H	C Language Executor application screen 4
	36H	C Language Executor application screen 5
	38H	CNC ID information screen
	39H	C Language Executor Task execution status monitor screen
	3DH	Parameter check sum screen
	3EH	Ethernet log [Embedded/PCMCIA] screen

Large classification	Small classification	Content
3	40H	Reference position setting with mechanical stopper screen
	41H	Data modification detection setting screen
	42H	CC-Link Remote device screen
	44H	Robot connection robot status screen
	45H	Robot connection setting screen
	46H	Power consumption monitoring screen
	47H	USB Maintenance screen
	49H	FL-net (2nd) screen
	4AH	Maintenance monitor screen
	4BH	Machining condition setting screen
	4CH	Machine state watch screen
	4DH	Machine state history screen
	4EH	Power consumption measurement setting screen
	4FH	PROFINET IO device function setting maintenance screen
	51H	EtherNet/IP Extended screen
	52H	PMC axis control state display
	53H	PROFINET IO controller function setting screen
	54H	Machining short time setting screen
	55H	Scroll waiting M Code setting screen
4	01H	Alarm screen
	02H	Operator message screen
	03H	Alarm history screen
	04H	Operator message history screen
	06H	Embedded Ethernet port log screen
	07H	PCMCIA Ethernet card log screen
	08H	Ethernet log screen
	09H	FL-net (1st) communication history screen
	0BH	System alarm history screen
	0CH	USB log screen
	10H	DeviceNet Master HISTORY
	11H	DeviceNet Slave HISTORY
	12H	FL-net (2nd) communication history screen
	13H	Trouble diagnosis guidance screen
	14H	Trouble diagnosis monitor screen
	15H	Trouble diagnosis parameter screen
	16H	PROFINET IO device function log screen
	18H	EtherNet/IP Extended log screen
	19H	Safety communication log
	1AH	PROFINET IO controller function log screen
	32H	C Language Executor application screen 1
	33H	C Language Executor application screen 2
	34H	C Language Executor application screen 3
	35H	C Language Executor application screen 4
	36H	C Language Executor application screen 5
5	01H	Graphic parameter screen
	02H	Tool path graphic screen
	06H	Graphic parameter screen of the dynamic graphic
	07H	Path graphic screen of the dynamic graphic
	08H	Animation graphic screen of the dynamic graphic
	09H	Path graphic (tool position) screen of the dynamic graphic
	32H	C Language Executor application screen 1
	33H	C Language Executor application screen 2

Large classification	Small classification	Content
5	34H	C Language Executor application screen 3
	35H	C Language Executor application screen 4
	36H	C Language Executor application screen 5
6	01H	Conversational macro 1 screen
	02H	Conversational macro 2 screen
	03H	Conversational macro 3 screen
	08H	PMC C Language board application screen
	32H	C Language Executor application screen 1
	33H	C Language Executor application screen 2
	34H	C Language Executor application screen 3
	35H	C Language Executor application screen 4
	36H	C Language Executor application screen 5
7	32H	C Language Executor application screen 1
	33H	C Language Executor application screen 2
	34H	C Language Executor application screen 3
	35H	C Language Executor application screen 4
	36H	C Language Executor application screen 5
8	01H	Main Menu Running screen
	02H	Main Menu Edit screen
	03H	Main Menu Setup screen
	04H	Main Menu Maintenance screen
	05H	Main Menu History screen
	32H	C Language Executor application screen 1
	33H	C Language Executor application screen 2
	34H	C Language Executor application screen 3
	35H	C Language Executor application screen 4
	36H	C Language Executor application screen 5

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F523	SCRNL7	SCRNL6	SCRNL5	SCRNL4	SCRNL3	SCRNL2	SCRNL1	SCRNL0
F524	SCRNS7	SCRNS6	SCRNS5	SCRNS4	SCRNS3	SCRNS2	SCRNS1	SCRNS0

17 EMBEDDED ETHERNET FUNCTION

17.1 EMBEDDED ETHERNET PORT AND PCMCIA ETHERNET CARD

The embedded Ethernet function can be used by selecting one of two types of devices: the embedded Ethernet port and PCMCIA Ethernet card.
A selection can also be made to stop the embedded Ethernet function.
The PCMCIA Ethernet card is to be inserted into the memory card slot for temporary communication.

⚠ **CAUTION**

1 When using the embedded Ethernet function for the first time, set an IP address and other items carefully as instructed by the network administrator, then perform a sufficient communication test.
Note that an incorrect IP address or other setting may cause a communication failure on the entire network or system error in the CNC.

2 A unit such as a PC situated in the same network can increase the communication processing load on the CNC even if the unit is not communicating with the CNC.
Avoid connecting the CNC to a factory-wide network. Use a router or the like to separate the network including the CNC from the other networks.

3 The PCMCIA Ethernet card is inserted into a memory card slot, with a part of the card left uninserted. When using the PCMCIA Ethernet card, take great care not to damage the card by hitting the protruding part of the card.
When the card becomes unnecessary, remove the card immediately, in order to prevent any damage to the card.

NOTE

1 Use the PCMCIA Ethernet card designated by FANUC. General Ethernet cards available on the market cannot be used.
2 The PCMCIA Ethernet card is used for FANUC LADDER-III or a SERVO GUIDE.
3 Use the PCMCIA Ethernet card just for temporary communication as described above. Avoid using the card for continuous communication.

Related parameters

	#7	#6	#5	#4	#3	#2	#1	#0
14880								ETH

[Input type] Parameter input
[Data type] Bit

#0 ETH The embedded Ethernet function (a built-in port or PCMCIA LAN card) is:
0: Used.
1: Not used.

17.2 SETTING UP THE EMBEDDED ETHERNET FUNCTION

This section describes the setting of parameters for the embedded Ethernet function.

17.2.1 Setting of the FOCAS2/Ethernet Function

This subsection describes the settings required to operate the FOCAS2/Ethernet function.

> ⚠ **WARNING**
> When using the FOCAS2/Ethernet function, thoroughly confirm the content of "GENERAL WARNINGS FOR CNC APPLICATION DEVELOPMENT" in "SAFETY PRECAUTIONS" at the beginning of this manual.
> If the machine is operated without confirmation of above explanation, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

Notes on using the FOCAS2/Ethernet function for the first time

> **NOTE**
> 1 When running user's original application software created by using the FOCAS2/Ethernet function, use the embedded Ethernet port.
> 2 The FOCAS2/Ethernet function allows up to five FOCAS2/Ethernet clients to be connected to one CNC.
> 3 Concurrent access by multiple applications or personal computers may overload the CNC, reducing the communication speed.
> 4 For the FOCAS2/Ethernet functions, the machine tool builder or end user can create desired software operated on each personal computer using the FOCAS2 library software.
> Communication timing and other factors may vary depending on the personal computer environment. Before starting communication, completely confirm that there is no problem with the entire system including communication with the personal computer.

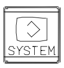
Notes on using the CNC Screen Display function for the first time

> **NOTE**
> With the CNC screen display functions, up to 1 CNC screen display function client can be connected to one CNC.

17.2.1.1 Operation on the FOCAS2/Ethernet setting screen

On the Ethernet parameter setting screen, set the parameters for operating the FOCAS2/Ethernet function.

Procedure

1 Press the function key 

.

2 Soft keys [EMBED] and [PCMCIA LAN] appear.
(When there is no soft keys, press the continue key.)

3 To display the Ethernet Setting screen for the embedded Ethernet port or the PCMCIA Ethernet card, press soft key [EMBED] or [PCMCIA LAN], respectively.

4 Press soft keys [COMMON] and [FOCAS2] and then enter parameters for the items that appear.

NOTE
1. The parameters for the embedded Ethernet port and the parameters for the PCMCIA Ethernet card are independent of each other.
2. The settings of the FOCAS2/Ethernet function for the PCMCIA Ethernet card are made when a connection to the SERVO GUIDE and FANUC LADDER-III is established.

COMMON screen (BASIC)

Press soft key [COMMON]. The COMMON screen (BASIC) (Fig. 17.2.1.1 (a)) is displayed.

Fig. 17.2.1.1 (a) COMMON screen (BASIC)

Setting items

Item	Description
IP ADDRESS	Specify the IP address of the embedded Ethernet. (Example of specification format: "192.168.0.100")
SUBNET MASK	Specify a mask address for the IP addresses of the network. (Example of specification format: "255.255.255.0")
ROUTER IP ADDRESS	Specify the IP address of the router. Specify this item when the network contains a router. (Example of specification format: "192.168.0.253")
DHCP CLIENT	The value of parameter No.14880#6(DHC) related to the DHCP client function is displayed, and the setting is also possible. For details, see Subsection, "Setting Up the DNS/DHCP Function".

Display items

Item	Description
MAC ADDRESS	Embedded Ethernet MAC address
AVAILABLE DEVICE	Enabled device of the embedded Ethernet. Either the embedded Ethernet port or the PCMCIA Ethernet card is displayed.

FOCAS2 screen

Press soft key [FOCAS2]. The FOCAS2 screen (Fig. 17.2.1.1(b)) is displayed.

Fig. 17.2.1.1 (b) FOCAS2 screen

Setting items

Item	Description
PORT NUMBER (TCP)	Specify a port number to be used with the FOCAS2/Ethernet function. The valid input range is 5001 to 65535.
PORT NUMBER (UDP)	Set this item to 0.
TIME INTERVAL	Set this item to 0.

Initial setting of the PCMCIA Ethernet card

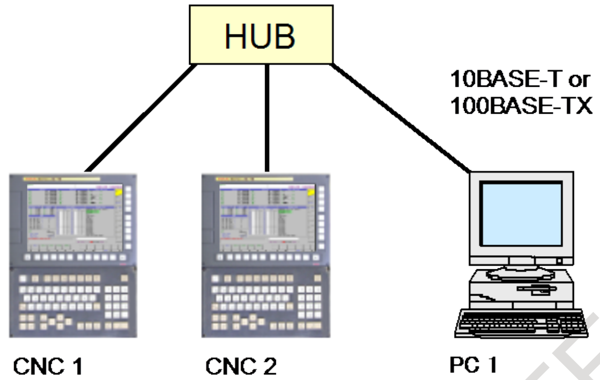
The PCMCIA Ethernet card is factory-set to the following standard setting values, for ease of connection with a SERVO GUIDE or FANUC LADDER-III.

IP ADDRESS	: 192.168.1.1
SUBNET MASK	: 255.255.255.0
ROUTER IP ADDRESS	: None
PORT NUMBER (TCP)	: 8193
PORT NUMBER (UDP)	: 0
TIME INTERVAL	: 0

If a specified IP address is changed to a blank (space), the specified setting is reset to the standard setting value.
The embedded Ethernet port does not have a standard setting value.

17.2.1.2 Example of setting the FOCAS2/Ethernet function

The following shows a setting example required for the FOCAS2/Ethernet function to operate. (when Windows 10 is used as the OS for the personal computer).
In this example, one personal computer is connected to two CNCs through FOCAS2/Ethernet.



	CNC 1	CNC 2
IP address	192.168.0.100	192.168.0.101
Subnet mask	255.255.255.0	255.255.255.0
Router IP address	None	None
TCP port number	8193	8193
UDP port number	0	0
Time interval	0	0

The Ethernet parameter screen is used for setting.

		PC 1
IP address		192.168.0.200
Subnet mask		255.255.255.0
Default gateway		None
CNC 1	NC IP address	192.168.0.100
	NC TCP port number	8193
CNC 2	NC IP address	192.168.0.101
	NC TCP port number	8193

"Internet Protocol Version 4 (TCP/IPv4) Properties" of the personal computer (Windows 10) is used for setting.

The arguments of the data window library function cnc_allclibhndl3 are used for setting.

17.2.2 Setting of the FTP File Transfer Function

This section describes the settings required for the FTP file transfer function to operate using the embedded Ethernet function.

Notes on using the FTP file transfer function for the first time

NOTE
1 When using the FTP file transfer function, use the embedded Ethernet port.
2 The number of FTP communications to which one CNC can be connected using the FTP file transfer function is one.
3 Before communicating with the host computer using the FTP file transfer function, completely confirm that there is no problem with the entire system including communication with the personal computer because the personal computer environment may affect communication operation.
4 To operate the host computer as the FTP server, FTP server software must be operated on the host computer.
5 When a lot of files are registered in the host computer, the display processing of host file list display might become slow. In this case, divide the folder that registers the files.

17.2.2.1 Operation on the FTP file transfer setting screen

On the Ethernet setting screen, set the parameters for operating the FTP file transfer function.

Procedure

1 Press the function key SYSTEM.
2 Soft keys [EMBED] appear.
(When there is no soft keys, press the continue key.)
3 By pressing the [EMBED] soft key, the Ethernet Setting screen for the embedded Ethernet port is displayed.
4 Press soft keys [COMMON] and [FTP TRANS] and then enter parameters for the items that appear.

NOTE
The parameters for the embedded Ethernet port and the parameters for the PCMCIA Ethernet card are independent of each other.
If the [PCMCIA LAN] soft key is pressed, the PCMCIA Ethernet card can be set up. However, the card setup is carried out for maintenance and is not necessary usually.

COMMON screen (BASIC)

Press soft key [COMMON]. The COMMON screen (BASIC) (Fig. 17.2.2.1(a)) is displayed.

Fig. 17.2.2.1 (a) COMMON screen (BASIC)

Setting items

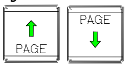
Item	Description
IP ADDRESS	Specify the IP address of the embedded Ethernet. (Example of specification format: "192.168.0.100")
SUBNET MASK	Specify a mask address for the IP addresses of the network. (Example of specification format: "255.255.255.0")
ROUTER IP ADDRESS	Specify the IP address of the router. Specify this item when the network contains a router. (Example of specification format: "192.168.0.253")
DHCP CLIENT	The value of parameter No.14880#6(DHC) related to the DHCP client function is displayed, and the setting is also possible. For details, see Subsection, “Setting Up the DNS/DHCP Function”.

Display items

Item	Description
MAC ADDRESS	Embedded Ethernet MAC address
AVAILABLE DEVICE	Enabled device of the embedded Ethernet. Either the embedded Ethernet port or the PCMCIA Ethernet card is displayed.

FTP transfer screen (CONNECT1, CONNECT2, CONNECT3)

1 Press soft key [FTP TRANS]. The FTP transfer screen (Fig. 17.2.2.1(b) and (c)) is displayed.

2 Page keys  can be used to make settings for the three host computers for connection destinations 1 to 3.

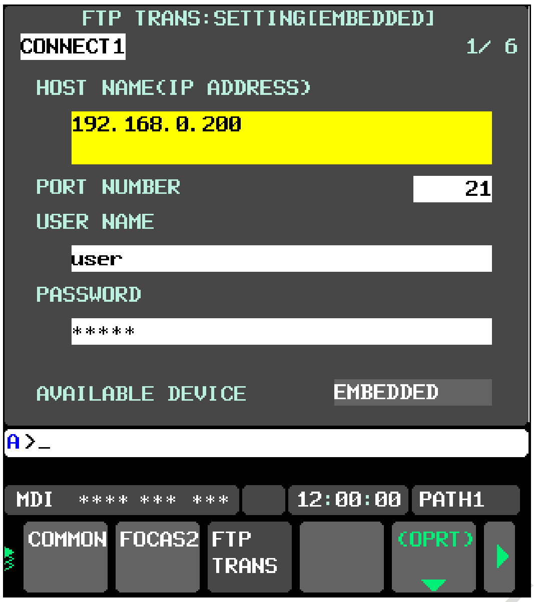


Fig. 17.2.2.1 (b) FTP transfer screen (1st page)

Fig. 17.2.2.1 (c) FTP transfer screen (2nd page)

Item	Description
HOST NAME	Specify the IP address of the host computer. (Example of specification format: "192.168.0.200")
PORT NUMBER	Specify a port number to be used with the FTP file transfer function. An FTP session is used, so that "21" is to be specified usually.
USER NAME	Specify a user name to be used for logging in to the host computer with FTP. (Up to 31 characters can be specified.)
PASSWORD	Specify a password for the user name specified above. (Up to 31 characters can be specified.) Be sure to set a password.
LOGIN FOLDER	Specify a work folder to be used when logging in to the host computer. (Up to 127 characters can be specified.) If nothing is specified, the home folder specified in the host computer becomes the log-in folder.

Operation

Select a destination.

1 Pressing the [(OPRT)] soft key causes soft key [HOST SELECT] to be displayed. Pressing this soft key causes soft keys [CONECT 1], [CONECT 2], and [CONECT 3] to be displayed. (Fig. 17.2.2.1(d))

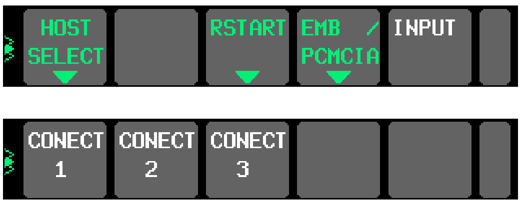
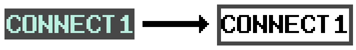


Fig. 17.2.2.1 (d)

2 Depending on the host computer to be connected, press soft key [CONECT 1], [CONECT 2], or [CONECT 3]. CONNECT 1, 2, or 3 is highlighted in the screen title field. The computer corresponding to the highlighted destination is selected as the target computer to be connected.



When CONNECT 1 is selected

17.2.2.2 Related parameters

The parameters related to the FTP file transfer function are described below.

0020	I/O CHANNEL : Input/output device selection, or interface number for a foreground input device

[Input type] Setting input
[Data type] Byte
[Valid data range] 9 : Select the embedded Ethernet as the input/output device.

For embedded Ethernet port

	#7	#6	#5	#4	#3	#2	#1	#0
14880							PCH	

[Input type] Parameter input
[Data type] Bit

#1 PCH When communication based on the FTP file transfer function starts, an FTP server presence check based on PING is:
0: Made.
1: Not made.

> **NOTE**
> Usually, set this parameter to 0 (to make a check).
> If this parameter is set to 1 (not to make an FTP server presence check based on PING), several tens of seconds may be required until an error is recognized when no FTP server is present on the network.
> Mainly for security, a personal computer may be set to ignore the PING command. When communicating with such a personal computer, set this parameter to 1 (not to make an FTP server presence check based on PING).

	#7	#6	#5	#4	#3	#2	#1	#0
14883	PSV							

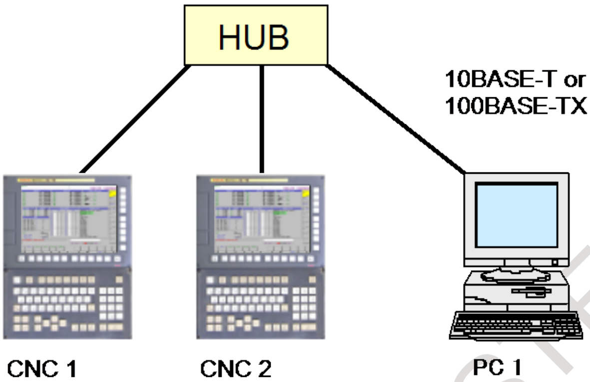
[Input type] Parameter input
[Data type] Bit

#7 PSV FTP client of FTP file transfer function:
0: Operates as the active mode.
1: Operates as the passive mode.

17.2.2.3 Example of setting the FTP file transfer function

The following shows a setting example required for the FTP file transfer function to operate.
(when Windows 10 is used as the OS for the personal computer).
In this example, one personal computer is connected to two CNCs through the FTP file transfer function.

- On Personal Computer 1, the FTP server function operates.
- On CNC 1 and CNC 2, the FTP client operates as the FTP file transfer function.



		CNC 1	CNC 2
IP address		192.168.0.100	192.168.0.101
Subnet mask		255.255.255.0	255.255.255.0
Router IP address		None	None
Connection host 1	Port number	21	21
	IP address	192.168.0.200	192.168.0.200
	User name	user	user
	Password	user	user
	Login DIR	None	None
Parameter No. 20		9	9

The Ethernet parameter screen is used for setting.

The parameter screen is used for setting.

	PC 1
IP address	192.168.0.200
Subnet mask	255.255.255.0
Default gateway	None
User name	user
Password	user
Login DIR	Default

"Internet Protocol Version 4 (TCP/IPv4) Properties" of the personal computer (Windows 10) is used for setting.

"Accounts" of the personal computer (Windows 10) is used for setting.

"Internet Information Service (IIS) Manager" of the personal computer (Windows 10) is used for setting.

17.2.3 Setting Up the DNS/DHCP Function

The DNS/DHCP function is set up by using the COMMON screen (DETAIL) and parameters.

17.2.3.1 Setting up DNS

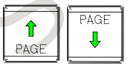
This subsection describes the procedure for setting up a DNS.

Procedure

1 Enable the DNS client function, with reference to "Related Parameters," which will be seen later.
2 Set up the DNS server of the host computer.
3 Connect the host computer on which the DNS server is working (hereafter referred to as a DNS server), reboot the CNC, then press function key 

.
4 Press soft keys [EMBED] and [COMMON] in that order. The COMMON screen (DETAIL) appears.
5 Enter the IP address of the DNS server in the corresponding DNS IP address field.

COMMON screen (DETAIL)

After pressing soft key [COMMON], press either page key 

to call a desired COMMON screen (DETAIL) (Fig. 17.2.3.1(a)). Specify a DNS IP address.

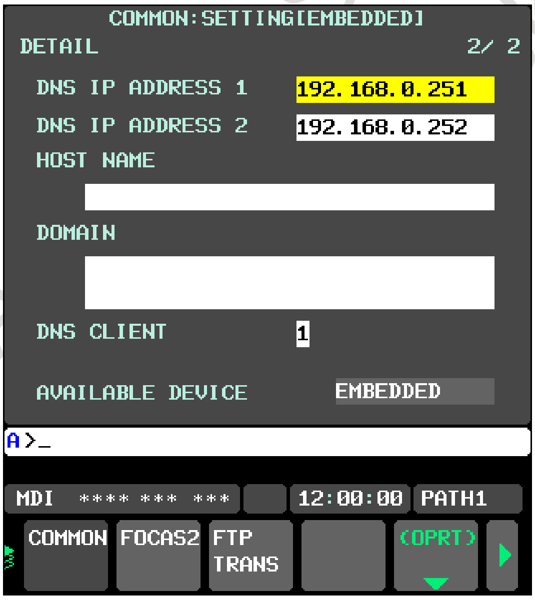


Fig. 17.2.3.1 (a) COMMON screen (DETAIL)

Display items

Item	Description
DNS IP ADDRESS 1, 2	Up to two DNS IP addresses can be specified. The CNC searches for the DNS server using DNS IP addresses 1 and 2 in that order.

17.2.3.2 Setting up DHCP

This subsection describes the procedure for setting up a DHCP.

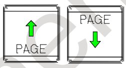
Procedure

1 Enable the DHCP client function, with reference to "Related Parameters," which will be seen later. Also this function can be enabled by setting “1” to the DHCP CLIENT in the COMMON screen (BASIC).
2 Set up the DHCP server of the host computer.
3 Connect the host computer on which the DHCP server is working (hereafter referred to as a DHCP server), reboot the CNC, then press function key 

.
4 Press soft keys [EMBED] and [COMMON] in that order. The COMMON screen appears.
5 If the DHCP client function of the CNC has been enabled and if the DHCP server is connected successfully, the DHCP server automatically specifies the following items.
 - IP ADDRESS
 - SUBNET MASK
 - ROUTER IP ADDRESS
 - DNS IP ADDRESS
 - DOMAIN

 If the DHCP server cannot be connected, "DHCP ERROR" is displayed in each field.
6 If the DNS client function has also been enabled and if the DHCP server and the DNS server work together (if the DNS server supports dynamic DNS), enter a host name.

COMMON screen (BASIC and DETAIL)

After pressing soft key [COMMON], press either page key 

to call desired Ethernet common setting screens (BASIC and DETAIL).
If the DHCP server is connected successfully and if the setting data can be obtained, the screens are displayed as shown Fig. 17.2.3.2(a).

Fig. 17.2.3.2 (a) When the DHCP server is connected successfully

If the host name is not specified, the CNC automatically assigns a host name in the "NC-<MAC-address>" format.

Example of automatically assigned host name

If the DHCP server cannot be connected, the screens are displayed as shown Fig. 17.2.3.2(b).

Fig. 17.2.3.2 (b) When the DHCP server cannot be connected

Check items

Item	Description
IP ADDRESS SUBNET MASK ROUTER IP ADDRESS DNS IP ADDRESS 1,2 DOMAIN	If the DHCP server is connected successfully, the items obtained from the DHCP server are displayed. If the DHCP server cannot be connected, "DHCP ERROR" is displayed.

Setting items

Item	Description
HOST NAME	Enter the host name of the CNC. If a DHCP server and a DNS server work together, the DHCP server notifies the DNS server of this host name. If the host name is left blank, a host name is automatically assigned in the "NC-<MAC-address>" format. Example of automatically assigned host name: NC-080019000001
DNS CLIENT	The value of parameter No.14880#5(DNS) related to the DNS client function is displayed, and the setting is also possible. See “Related Parameters”, which will be seen later.

Display items

Item	Description
MAC ADDRESS	MAC address of embedded Ethernet

17.2.3.3 Related parameters

For embedded Ethernet port

	#7	#6	#5	#4	#3	#2	#1	#0
14880		DHC	DNS					

[Input type] Parameter input
[Data type] Bit

#5 DNS With a built-in port, the DNS client function is:
0: Not used.
1: Used.
This value is also displayed to the DNS CLIENT in the COMMON screen (DETAIL), and the setting is also possible.

#6 DHC With a built-in port, the DHCP client function is:
0: Not used.
1: Used.
This value is also displayed to the DHCP CLIENT in the COMMON screen (BASIC), and the setting is also possible.

A change in these parameters becomes effective after the power is turned off and on or after the embedded Ethernet function is restarted.

17.2.4 Setting of the CNC Screen Display Function

CNC Screen Display function is a software that allows performing display and operation similar to a CNC on the Windows PC. Refer to "CNC Screen Display Function OPERATOR’S MANUAL (B-63164EN)" for the method of use.

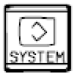
NOTE

1 The screen update interval of CNC Screen Display function is slower than that of a CNC-dedicated display unit. Therefore, use this function as an auxiliary display unit.
2 When high-load machining such as high-speed and high-precision machining is performed or the functions that require the CPU power are specified, the screen update interval might be slower. Even if a higher-performance personal computer is used, the screen update interval is not improved.
Therefore, the CNC Screen Display function on Embedded Ethernet might not show the full performance according to the system configuration of the machine.
3 On Embedded Ethernet, when other network functions (FOCAS2/Ethernet function etc.) are used simultaneously with CNC Screen Display function, it affects the screen update interval. Therefore, do not use other network functions at the same time on Embedded Ethernet when using CNC Screen Display function.

17.2.4.1 Operation on the Setting screen

This section describes the setting screen for operating the CNC Screen Display function.

Procedure

1 Press the function key 

.

2 Soft key [EMBED PORT] appears. (When there is not this soft keys, press the continue key.)

3 Press soft key [EMBED PORT] to display the Ethernet Setting screen.

4 Press soft keys [COMMON] and [FOCAS2], and then enter parameters for the items that appear.

COMMON screen (BASIC)

Press soft key [COMMON] to display the COMMON screen (BASIC).

Fig. 17.2.4.1 (a) COMMON screen (BASIC)

Setting items

Item	Description
IP ADDRESS	Specify the IP address of the embedded Ethernet. (Example of specification format: "192.168.0.100")
SUBNET MASK	Specify a mask address for the IP addresses of the network. (Example of specification format: "255.255.255.0")
ROUTER IP ADDRESS	Specify the IP address of the router. Specify this item when the network contains a router. (Example of specification format: "192.168.0.253")
DHCP CLIENT	The value of parameter No.14880#6(DHC) related to the DHCP client function is displayed, and the setting is also possible. For details, see Subsection, "Setting Up the DNS/DHCP Function".

Display items

Item	Description
MAC ADDRESS	Embedded Ethernet MAC address
AVAILABLE DEVICE	Enabled device of the embedded Ethernet. Either the embedded Ethernet port or the PCMCIA Ethernet card is displayed.

FOCAS2 screen

Press soft key [FOCAS2] to display the FOCAS2 screen.

Fig. 17.2.4.1 (b) FOCAS2 screen

Setting items

Item	Description
PORT NUMBER (TCP)	Specifies the port No. to be used by the CNC screen Display function (FOCAS2/Ethernet function), within a range of 5001 to 65535.
PORT NUMBER (UDP)	Set this item to 0.
TIME INTERVAL	Set this item to 0.

17.2.5 Setting of the Machine Remote Diagnosis package

Machine Remote Diagnosis package is a tool designed for the machine tool builder to provide remote support of machines easily. Machine Remote Diagnosis function works on Embedded Ethernet or Fast Ethernet board in order to communicate with Machine Remote Diagnosis package.
Refer to "Machine Remote Diagnosis Package OPERATOR'S MANUAL (B-63734EN)" for the method of use.

> **NOTE**
> Refer to the notes of “Setting of the CNC Screen Display function” when using CNC Screen Display function for MRDP (Machine Remote Diagnosis Package).The notes are applied to this function, too.

17.2.5.1 Related NC parameters

When using Machine Remote Diagnosis package on Embedded Ethernet, the following parameters must be set.

0024	Setting of communication with the PMC ladder development tool

[Input type] Setting input
[Data type] Word
[Valid data range] 10: The high-speed interface (Ethernet) is used for PMC online editing.

	#7	#6	#5	#4	#3	#2	#1	#0
8706		MRD						

[Input type] Parameter input
[Data type] Bit

#6 MRD Machine Remote Diagnosis function is used:
0: On Embedded Ethernet.
1: On Fast Ethernet board.

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

	#7	#6	#5	#4	#3	#2	#1	#0
14880							PCH	

[Input type] Parameter input
[Data type] Bit
#1 PCH At the start of communication of the Data Server, FTP file transfer function, or machine remote diagnosis functions, checking for the presence of the server using PING is:
0: Performed.
1: Not performed.

> **NOTE**
> When connecting with the host computer (machine remote diagnosis accepting server) on Intranet, set 0.
> When the presence of the server using PING is not checked by setting 1, it may take several tens of seconds until an error is recognized due to no server on the network.
> When connecting with the host computer (machine remote diagnosis accepting server) via Internet, set 1.

	#7	#6	#5	#4	#3	#2	#1	#0
14882								ERD

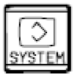
[Input type] Parameter input
[Data type] Bit

> **NOTE**
> Re-setting this parameter requires turning the power off and on again or restarting the embedded Ethernet interface.

#0 ERD On Embedded Ethernet, Machine Remote Diagnosis is:
0: Not used.
1: Used.

17.2.5.2 Operation on the Setting screen

Procedure

1 Press the function key .

2 Soft key [EMBED PORT] appears. (When there is not this soft keys, press the continue key.)

3 Press soft key [EMBED PORT] to display the Ethernet Setting screen.

4 Press soft keys [COMMON], [FOCAS2], and [REMOTE DIAG] and then enter parameters for the items that appear.

> **NOTE**
> 1 The parameters for the embedded Ethernet port and the parameters for the PCMCIA Ethernet card are independent of each other.
> 2 If the [PCMCIA LAN] soft key is pressed, the PCMCIA Ethernet card can be set up. However, the card setup is carried out for maintenance and is not necessary usually.

COMMON screen (BASIC)

Press soft key [COMMON] to display the COMMON screen (BASIC).

Fig. 17.2.5.2 (a) COMMON screen (BASIC)

Setting items

Item	Description
IP ADDRESS	Specify the IP address of the embedded Ethernet. (Example of specification format: "192.168.0.100")
SUBNET MASK	Specify a mask address for the IP addresses of the network. (Example of specification format: "255.255.255.0")
ROUTER IP ADDRESS	Specify the IP address of the router. Specify this item when the network contains a router. (Example of specification format: "192.168.0.253")
DHCP CLIENT	The value of parameter No.14880#6(DHC) related to the DHCP client function is displayed, and the setting is also possible. For details, see Subsection, "Setting Up the DNS/DHCP Function".

Display items

Item	Description
MAC ADDRESS	Embedded Ethernet MAC address
AVAILABLE DEVICE	Enabled device of the embedded Ethernet. Either the embedded Ethernet port or the PCMCIA Ethernet card is displayed.

FOCAS2 screen

Press soft key [FOCAS2] to display the FOCAS2 screen.

Fig. 17.2.5.2 (b) FOCAS2 screen

Setting items

Item	Description
PORT NUMBER (TCP)	Specifies the port No. to be used by the machine remote diagnosis functions (FOCAS2/Ethernet functions), within a range of 5001 to 65535.
PORT NUMBER (UDP)	Set this item to 0.
TIME INTERVAL	Set this item to 0.

MACHINE REMOTE DIAG screen (COMMON)

Press soft key [REMOTE DIAG] to display the MACHINE REMOTE DIAG screen (COMMON).

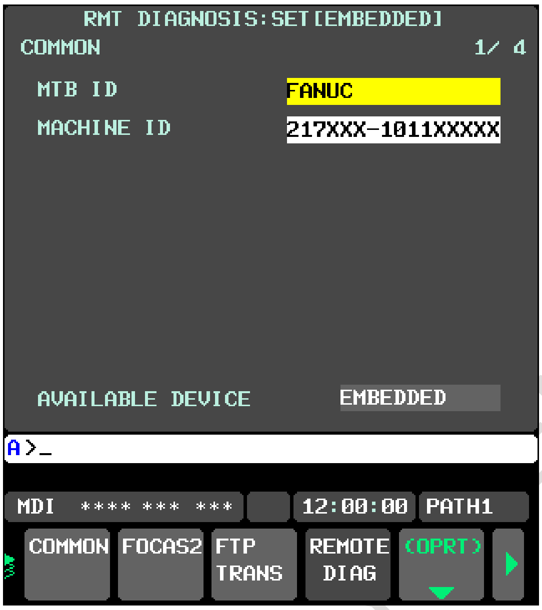


Fig. 17.2.5.2 (c) Machine remote diagnosis screen (COMMON)

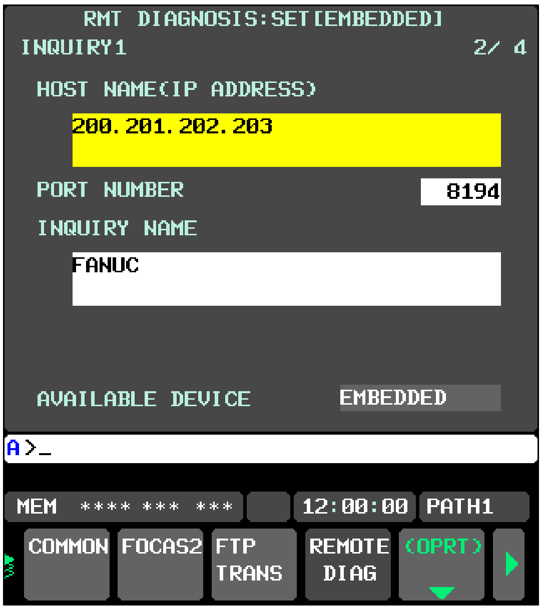
Setting items

Item	Description
MTB ID	This information is required by the machine remote diagnosis package to confirm that the diagnosis request is issued from a machine manufactured by the machine tool builder. The MTB identification information on the diagnosis accepting server of the machine remote diagnosis package can be set to accept diagnosis requests only from the machines manufactured by the machine tool builder. (Example of specification format: "FANUC")
MACHINE ID	Information required by the machine remote diagnosis package to identify the machine under diagnosis (Example of specification format: "217xxx-1011xxxxx")

MACHINE REMOTE DIAG screen (INQUIRY1, INQUIRY2, INQUIRY3)

Press soft key [RMTDIAG] to display the MACHINE REMOTE DIAG screen.

By using page keys PAGE PAGE, the three host computers at inquiry destinations 1, 2, and 3 can be set.



(When the DNS function is disabled)

(When the DNS function is enabled)

Fig. 17.2.5.2 (d) Machine Remote Diagnosis screens (INQUIRY1)

Setting item

Item	Description
HOST NAME	Specify the IP address of the host computer (machine remote diagnosis accepting server) when the DNS function is disabled. (Example of specification format: "200.201.202.203") Specify the host name of the host computer (machine remote diagnosis accepting server) when the DNS function is enabled. (You can specify up to 63 characters.) (Example of specification format: "RMTDIAG.FANUC.CO.JP")
PORT NUMBER	Specify a port number. Usually, specify "8194" because the machine remote diagnosis functions are used.
INQUIRY NAME	Specify information for identifying the host computer (machine remote diagnosis accepting server). (You can specify up to 63 characters.) (Example of specification format: "FANUC")

17.2.5.3 Controlling Machine Remote Diagnosis function from PMC

You can use signals from the PMC to control the start and forced termination of the machine remote diagnosis functions and post the status of the machine remote diagnosis functions and error numbers to the PMC ladder.

Signals

No.	#7	#6	#5	#4	#3	#2	#1	#0
G0141			DIAREQ	DIASTP		INQU2	INQU1	INQU0

DIAREQ <G0141.5>
[Name] Signal to request machine remote diagnosis
[Classification] Input signal
[Function] Requests the start of machine remote diagnosis.
[Operation] When this signal is set to "1", it requests the start of machine remote diagnosis to the inquiry destination according to the signals indicating the number of the inquiry destination (INQU0<G0141.0> to INQU2<G0141.2>). When the acceptance completion signal (RMTEND<F0083.6>) or acceptance reject signal (RMTCAN<F0083.7>) is set to "1", this signal is set to "0".

DIASTP <G0141.4>
[Name] Signal to request machine remote diagnosis cancellation
[Classification] Input signal
[Function] Requests the forced termination of machine remote diagnosis.
[Operation] When this signal is set to "1", it requests forced termination to the machine remote diagnosis accepting server. When the completion signal for machine remote signal cancel acceptance (RMTCLS<F0082.3>) is set to "1", this signal is set to "0".

INQU2 <G0141.2>
INQU1 <G0141.1>
INQU0 <G0141.0>
[Name] Inquiry number select signals
[Classification] Input signal
[Function] Inquiry destination for which to start machine remote diagnosis
[Operation] Select an item from the table below as an inquiry destination for which to start machine remote diagnosis.

INQU2	INQU1	INQU0	Status
0	0	0	No selection
0	0	1	Inquiry destination 1
0	1	0	Inquiry destination 2
0	1	1	Inquiry destination 3

No.	#7	#6	#5	#4	#3	#2	#1	#0
F0082					RMTCLS			

RMTCLS <F0082.3>
[Name] Completion signal for machine remote diagnosis cancel acceptance
[Classification] Output signal
[Function] Notifies that a request to cancel machine remote diagnosis has been accepted.
[Output condition] When machine remote diagnosis is canceled after the signal to request machine remote diagnosis cancellation (DIASTP<G0141.4>) is set to "1", this signal is set to "1". When the signal to request machine remote diagnosis cancellation (DIASTP<G0141.4>) is set to "0", this signal is set to "0".

No.	#7	#6	#5	#4	#3	#2	#1	#0
F0083	RMTCAN	RMTEND	DIAST5	DIAST4	DIAST3	DIAST2	DIAST1	DIAST0

RMTCAN <F0083.7>

[Name] Reject signal for machine remote diagnosis acceptance

[Classification] Output signal

[Function] Notifies that a machine remote diagnosis request has been rejected.

[Output condition] When the signal to request machine remote diagnosis (DIAREQ<G0141.5>) is set to "1", a request to start machine remote diagnosis is issued to the machine remote diagnosis accepting server. When the server rejects the request, this signal is set to "1". When the signal to request machine remote diagnosis (DIAREQ<G0141.5>) is set to "0", this signal is set to "0".

RMTEND <F0083.6>

[Name] Completion signal for machine remote diagnosis acceptance

[Classification] Output signal

[Function] Notifies that a machine remote diagnosis request has been accepted by the machine remote diagnosis accepting server.

[Output condition] When the signal to request machine remote diagnosis (DIAREQ<G0141.5>) is set to "1", a request to start machine remote diagnosis is issued to the machine remote diagnosis accepting server. When the server accepts the request, this signal is set to "1". When the signal to request machine remote diagnosis (DIAREQ<G0141.5>) is set to "0", this signal is set to "0".

DIAST5 <F0083.5>
DIAST4 <F0083.4>
DIAST3 <F0083.3>
DIAST2 <F0083.2>
DIAST1 <F0083.1>
DIAST0 <F0083.0>

[Name] Notification signals for the machine remote diagnosis status

[Classification] Output signal

[Function] Report the status of machine remote diagnosis.

[Output condition] The status of machine remote diagnosis is reported as listed in the following table.

DIAST5	DIAST4	DIAST3	DIAST2	DIAST1	DIAST0	Description
0	0	0	0	0	0	No status
0	0	0	0	0	1	OPEN
0	0	0	0	1	0	OPENING
0	0	0	0	1	1	ACCEPTED
0	0	0	1	0	0	REFUSED
0	0	0	1	0	1	DIAGNOSING
0	0	0	1	1	0	DIAGNOSING
0	0	0	1	1	1	CLOSE
0	0	1	0	0	0	FORCE CLOSING
0	0	1	0	0	1	ERROR

No.	#7	#6	#5	#4	#3	#2	#1	#0
F0088	DIAER7	DIAER6	DIAER5	DIAER4	DIAER3	DIAER2	DIAER1	DIAER0

DIAER7 <F0088.7>
DIAER6 <F0088.6>
DIAER5 <F0088.5>
DIAER4 <F0088.4>
DIAER3 <F0088.3>
DIAER2 <F0088.2>
DIAER1 <F0088.1>
DIAER0 <F0088.0>

[Name] Notification signals for a machine remote diagnosis error number

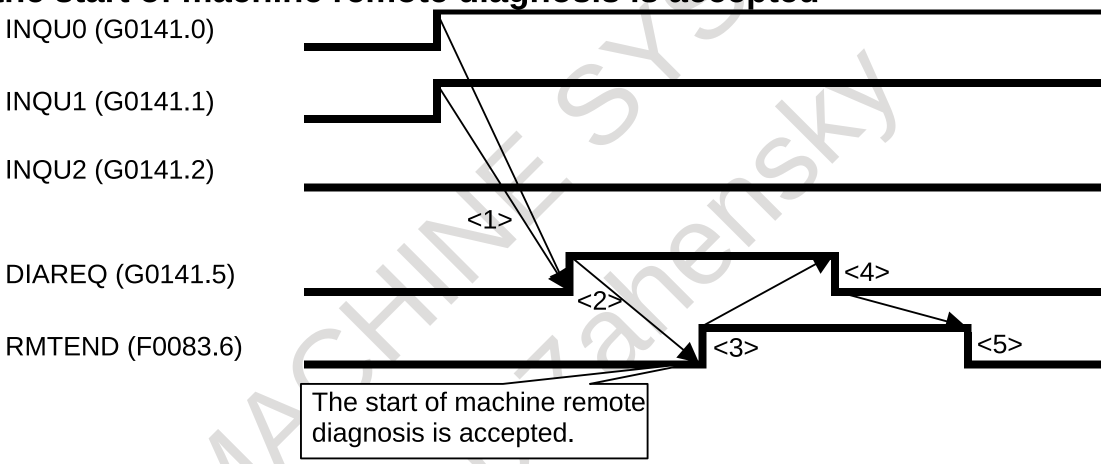
[Classification] Output signal

[Function] Report an error number of machine remote diagnosis.

[Output condition] These signals indicate an error number of machine remote diagnosis. The error number is 0 to 255 in binary format.

Signal Timing Charts

When the start of machine remote diagnosis is accepted



<1> Before the signal to request machine remote diagnosis (DIAREQ) is set to "1", an inquiry destination for which to start the remote diagnosis functions is selected using the inquiry destination number signals (INQU0, INQU1, and INQU2). In this example, "inquiry destination 3" is selected by setting INQU0 = 1, INQU1 = 1, and INQU2 = 0.

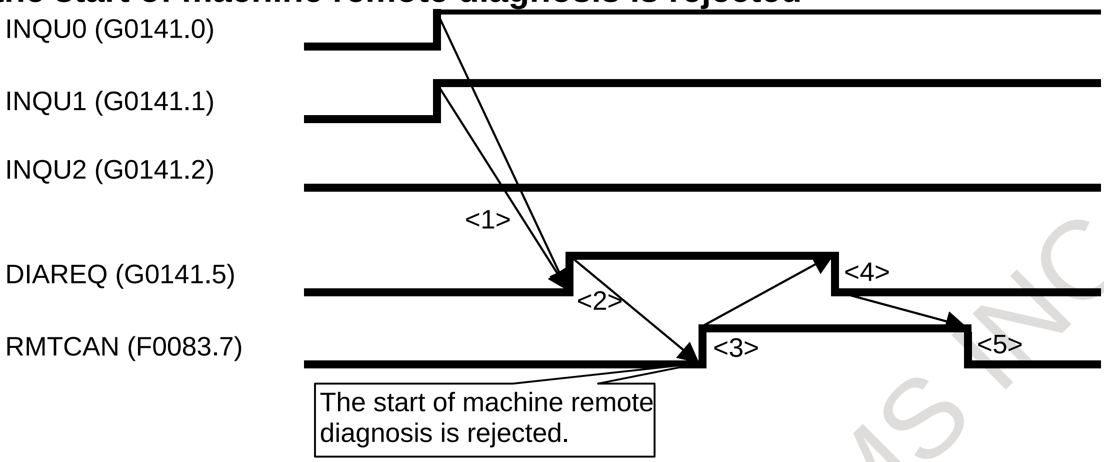
<2> The signal to request machine remote diagnosis (DIAREQ) is set to "1".

<3> When the machine remote diagnosis package accepts the request to start diagnosis, the completion signal for machine remote diagnosis acceptance (RMTEND) is set to "1".

<4> When the completion signal for machine remote diagnosis acceptance (RMTEND) is set to "1", the signal to request machine remote diagnosis (DIAREQ) is set to "0".

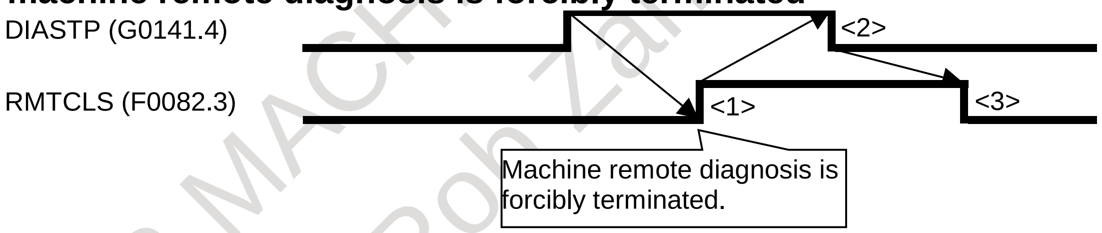
<5> When the signal to request machine remote diagnosis (DIAREQ) is set to "0", the completion signal for machine remote diagnosis acceptance (RMTEND) is set to "0".

When the start of machine remote diagnosis is rejected



<1> Before the signal to request machine remote diagnosis (DIAREQ) is set to "1", an inquiry destination for which to start the remote diagnosis functions is selected using the inquiry destination number signals (INQU0, INQU1, and INQU2). In this example, "inquiry destination 3" is selected by setting INQU0 = 1, INQU1 = 1, and INQU2 = 0.
<2> The signal to request machine remote diagnosis (DIAREQ) is set to "1".
<3> When the machine remote diagnosis package rejects the request to start diagnosis, the reject signal for machine remote diagnosis acceptance (RMTCAN) is set to "1".
<4> When the reject signal for machine remote diagnosis acceptance (RMTCAN) is set to "1", the signal to request machine remote diagnosis (DIAREQ) is set to "0".
<5> When the signal to request machine remote diagnosis (DIAREQ) is set to "0", the reject signal for machine remote diagnosis acceptance (RMTCAN) is set to "0".

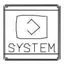
When machine remote diagnosis is forcibly terminated



<1> When the signal to request machine remote diagnosis cancellation (DIASTP) is set to "1", the completion signal for machine remote diagnosis cancel acceptance (RMTCLS) is set to "1".
<2> When the completion signal for machine remote diagnosis cancel acceptance (RMTCLS) is set to "1", the signal to request machine remote diagnosis cancellation (DIASTP) is set to "0".
<3> When the signal to request machine remote diagnosis cancellation (DIASTP) is set to "0", the completion signal for machine remote diagnosis cancel acceptance (RMTCLS) is set to "0" and the machine remote diagnosis functions are forcibly terminated.

17.2.5.4 Operating Machine Remote Diagnosis screen

Procedure

1 Press the function key .

2 Soft key [REMOTE DIAG] appear. (When there is no soft keys, press the continue key.)

3 Press soft key [REMOTE DIAG] to display the machine remote diagnosis screen.

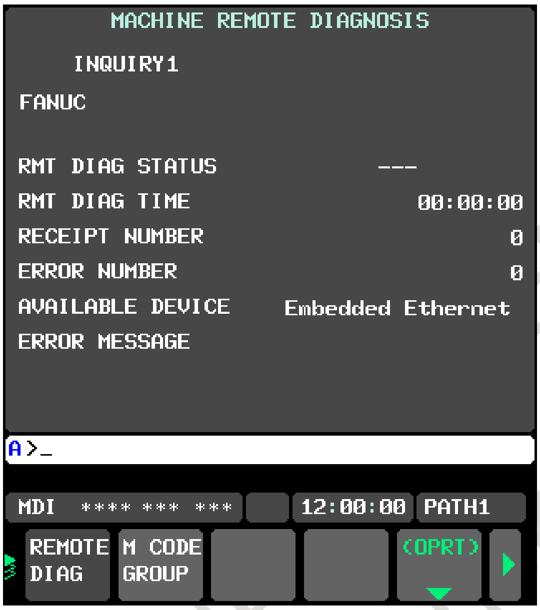


Fig. 17.2.5.4 (a) Machine Remote Diagnosis screen

4 Press soft key [(OPRT)] to display available soft keys.

Fig. 17.2.5.4 (b) Soft keys of Machine Remote Diagnosis screen

Display items

INQUIRY NUMBER

Displays the inquiry number indicating the machine remote diagnosis accepting server: "INQUIRY1," "INQUIRY2," or "INQUIRY3."

INQUIRY

Displays information for identifying the machine remote diagnosis accepting server.

RMT DIAG STATUS

Displays the status of machine remote diagnosis.

RMT DIAG TIME

Displays the time until the machine remote diagnosis status changes from "OPEN" to "CLOSE," "FORCE CLOSING," or "ERROR."
At each start of diagnosis, the time is accumulated from "00:00:00."

RECEIPT NUMBER

Displays the receipt number issued by the machine remote diagnosis accepting server.

ERROR NUMBER

Displays the number of an error which occurs in operation of the machine remote diagnosis functions.

AVAILABLE DEVICE
Displays the type of communication device for which the machine remote diagnosis functions can operate.

ERROR MESSAGE
Displays the message indicating an error which occurs in operation of the machine remote diagnosis functions.

Operation list

DIAG OPEN
Starts machine remote diagnosis.

DIAG CLOSE
Forcibly terminates machine remote diagnosis.

INQUIRY1
Selects inquiry destination 1.

INQUIRY2
Selects inquiry destination 2.

INQUIRY3
Selects inquiry destination 3.

Selecting an Inquiry Destination

Select an inquiry destination among inquiry destinations 1 to 3.

1. Press soft key [(OPRT)].
2. Press soft key [INQUIRY1] to select inquiry destination 1.
3. Similarly, press soft key [INQUIRY2] to select inquiry destination 2 and soft key [INQUIRY3] to select inquiry destination 3.

Starting Diagnosis

Start diagnosis.
1. Press soft key [(OPRT)].
2. Press soft key [DIAG OPEN] to send the start request to the machine remote diagnosis
3. When the machine remote diagnosis accepts the start request, the diagnosis begins.

Diagnosis status

Status	Description
---	No operation
OPEN	Soft key [DIAG OPEN] was pressed.
OPENING	An attempt is being made to connect the machine remote diagnosis accepting server.
ACCEPTED	The machine remote diagnosis accepting server accepted diagnosis.
REFUSED	The machine remote diagnosis accepting server rejected diagnosis.
DIAGNOSING	This message flashes in synchronization with data flowing on the communication line.
FORCE CLOSING	Soft key [DIAG CLOSE] was pressed. After the completion of forced termination processing, "CLOSE" is indicated in the RMT DIAG STATUS field.
CLOSE	The machine remote diagnosis accepting server terminated diagnosis.
ERROR	An error occurred on the communication line.

Error numbers and error messages

Number	Error message	Meaning and action to be taken
1	Diagnosis is busy	Soft key [DIAG OPEN] was pressed during diagnosis.
2	Router isn't alive	The IP address of the router may be invalid or the power to the router may be off. Check whether the IP address of the router is valid and whether the power to the router is on.
3	Receipt Server isn't alive	The IP address of the machine remote diagnosis accepting server may be invalid or the machine remote diagnosis accepting server may not be operating. Check whether the IP address of the machine remote diagnosis accepting server is valid and whether the machine remote diagnosis accepting server is operating.
4	System error	A system error occurred. Check the log messages on the ETHERNET LOG screen and contact FANUC.
5	Invalid Inquiry number.	A value outside the valid setting range may be set for the inquiry destination. Check whether the correct inquiry destination is set.
6	Invalid IP Address	Set the IP address according to the IP address specification format.
7	Invalid PORT number	A value outside the valid setting range may be set for the port number. Check whether the correct port number is set.
8	Invalid Router IP Address	Set the IP address of the router according to the IP address specification format.
9	Socket error	A communication error occurred due to a cause as listed below. Check the network wiring and anti-noise measures. → The network quality degraded, data could not be received from the personal computer with which to communicate, and the logical communication path was disconnected. → The software component on the personal computer with which to communicate forcibly disconnected the logical communication path. → The Ethernet cable was disconnected.
11	Invalid Request	An internal error related to machine remote diagnosis occurred in the CNC. Check the log messages on the ETHERNET LOG screen and contact FANUC.
12	Invalid Packet	An unrecognizable packet was received. Check the log messages on the ETHERNET LOG screen and contact FANUC.
13	Diagnosis was already stopped	Soft key [DIGA CLOSE] was pressed not during diagnosis.
17	Receive error	An attempt to receive data failed. See Number 9 and check the network wiring and anti-noise measures.
19	HeartBeat timeout	Communication with the machine remote diagnosis accepting server stopped. See Number 9 and check the network wiring and anti-noise measures.
20	HeartBeat error	An attempt was failed to send a heartbeat packet for machine remote diagnosis. See Number 9 and check the network wiring and anti-noise measures.
22	DNS error	An attempt was failed to connect the machine remote diagnosis accepting server using the DNS function. The IP address of the DNS server may be invalid or the power to the DNS server may be off. Check whether the IP address of the DNS server is valid and whether the power to the DNS server is on.

Forcibly Terminating Diagnosis

Forcibly terminate diagnosis.

1 Press soft key [(OPRT)].
2 Press soft key [DIAG CLOSE] to forcibly terminate diagnosis.

17.2.6 Setting of the Unsolicited Messaging Function

This subsection describes the setting required to operate the unsolicited messaging function with the embedded Ethernet function.

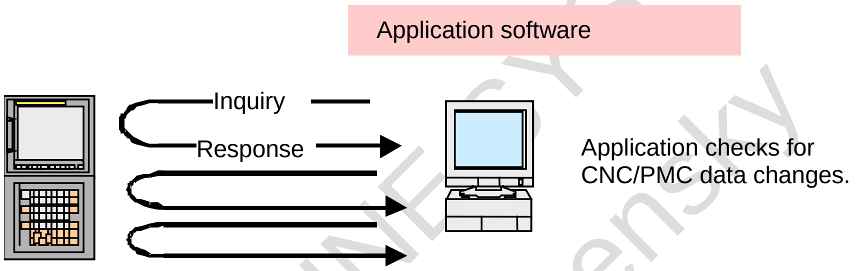
17.2.6.1 Overview

This subsection provides an overview of the unsolicited messaging function and describes the execution procedure.

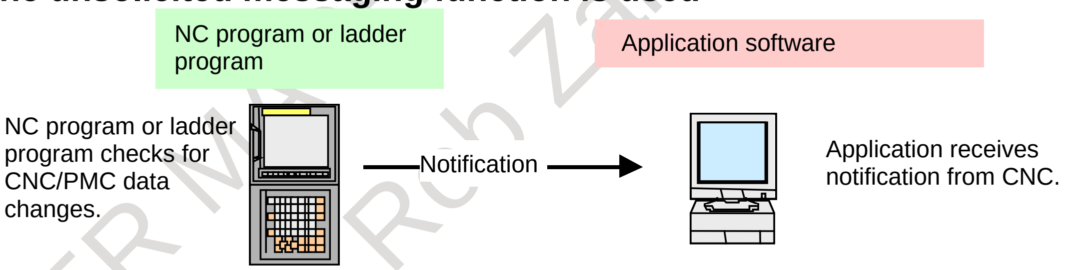
Overview of the unsolicited messaging function

An overview of the unsolicited messaging function is provided below.
With the unsolicited messaging function, the CNC transmits messages (CNC/PMC data) in an unsolicited manner to application software on the personal computer according to a command from an NC program or ladder program. By using this function, the need for application processing on the personal computer to periodically inquire about the state of the CNC can be eliminated.

When the conventional function is used



When the unsolicited messaging function is used



> **NOTE**
> The unsolicited messaging function is a part of the FOCAS2/Ethernet function.

Unsolicited messaging function execution procedure

The execution procedure for the unsolicited messaging function is described below.

1 Preparation on the personal compute
Create an application using the FOCAS2 function for the unsolicited messaging function and install the unsolicited message server on a personal computer. For information on creating an application with FOCAS2 functions for the unsolicited messaging function and installing the unsolicited message server, refer to Chapter 5, "UNSOLICITED MESSAGING FUNCTION" in "Data Window Library Specifications" that comes with the FOCAS1/2 library.

2 Preparation on the CNC
Create an NC program or ladder program for controlling unsolicited messaging.
For the method of creating an NC program or ladder program, see Subsection 17.2.6.6, "Execution methods".

3 Setting of the communication parameters for the unsolicited messaging function
To use the unsolicited messaging function, the following communication parameter settings are needed:
(1) Setting for using the FOCAS2/Ethernet function
(2) Setting of the parameters for the unsolicited messaging function
For (2), a choice can be made from two modes of setting: CNC mode for setting on the CNC screen and the PC mode for setting on the personal computer.
For the setting method of (1) and (2), see Subsection 17.2.6.2, "Setting of the FOCAS2/Ethernet function" through Subsection 17.2.6.5, "Setting on the personal computer".

4 Starting the NC program or ladder program
Start the NC program or ladder program created in step 2, "Preparation on the CNC". At this time, no unsolicited message is transmitted to the personal computer until step 5, "Starting the unsolicited messaging function", is executed.

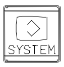
5 Starting the unsolicited messaging function
Execute the FOCAS2 function cnc_unsolicstart on the personal computer. This execution places the CNC in the state (named "Ready") where a transmission request from the NC program or ladder program is awaited. Each time a transmission request is made from the NC program or ladder program, an unsolicited message is automatically transmitted to the personal computer.

6 Ending the unsolicited messaging function
To end unsolicited message transmission, execute the FOCAS2 function cnc_unsolicstop on the personal computer. This execution places the CNC in the state (named "Not Ready") where no unsolicited message is transmitted even when a request for transmission is made from the NC program or ladder program.

17.2.6.2 Setting of the FOCAS2/Ethernet function

This subsection describes the setting of the FOCAS2/Ethernet function for operating the unsolicited messaging function.

Procedure

1 Enable the unsolicited messaging function according to "Related parameters" described later.
2 Start the CNC again then press function key 

.
3 Soft key [EMBED] is displayed. (Press the continuous menu key until the soft key is displayed.)
4 Press soft key [EMBED]. The Ethernet Setting screen for the embedded Ethernet port is displayed.
5 Press soft keys [COMMON] and [FOCAS2] then make settings on each screen.
6 Press soft key [UNSOLI MSG]. The Unsolicited Message screen is displayed. (Press the continuous menu key until the soft key [UNSOLI MSG] is displayed.) For details of the Unsolicited Message screen, see Subsections starting with Subsection 17.2.6.3, "Mode selection".

COMMON screen (BASIC)

Press soft key [COMMON]. The COMMON screen (BASIC) (Fig. 17.2.6.2(a)) is displayed.

Fig. 17.2.6.2 (a) COMMON screen (BASIC)

Setting items

Item	Description
IP ADDRESS	Specify the IP address of the embedded Ethernet. (Example of specification format: "192.168.0.100")
SUBNET MASK	Specify a mask address for the IP addresses of the network. (Example of specification format: "255.255.255.0")
ROUTER IP ADDRESS	Specify the IP address of the router. Specify this item when the network contains a router. (Example of specification format: "192.168.0.253")

Display items

Item	Description
MAC ADDRESS	Embedded Ethernet MAC address
AVAILABLE DEVICE	Enabled device of the embedded Ethernet. Either the embedded Ethernet port or the PCMCIA Ethernet card is displayed.

> **NOTE**
> Set page 2 (DETAIL screen) of the COMMON screen when using the DNS/DHCP function. For details, see Subsection 17.2.3, "Setting Up the DNS/DHCP Function".

COMMON screen (DETAIL)

When using the DNS client function, press soft key [COMMON] then press page key . The COMMON screen (DETAIL) (Fig. 17.2.6.2(b)) is displayed. Set the DNS IP address setting items.

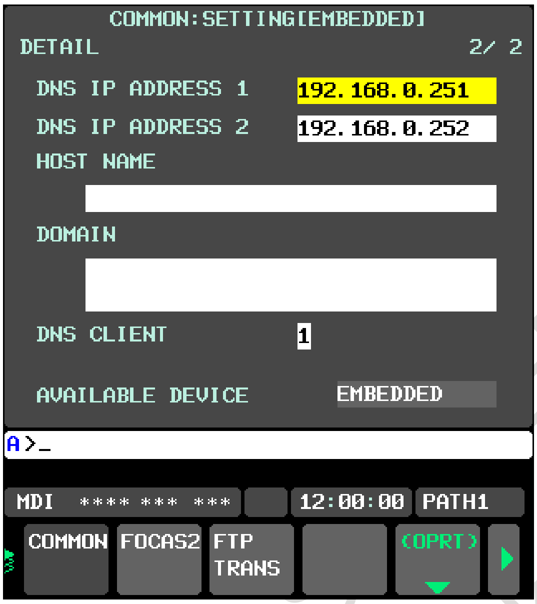


Fig. 17.2.6.2 (b) COMMON screen (DETAIL)

Display items

Item	Description
DNS IP ADDRESS 1, 2	Up to two DNS IP addresses can be specified. The CNC searches for the DNS server using DNS IP addresses 1 and 2 in that order.

FOCAS2 screen

Press soft key [FOCAS2]. The FOCAS2 screen (Fig. 17.2.6.2(c)) is displayed.

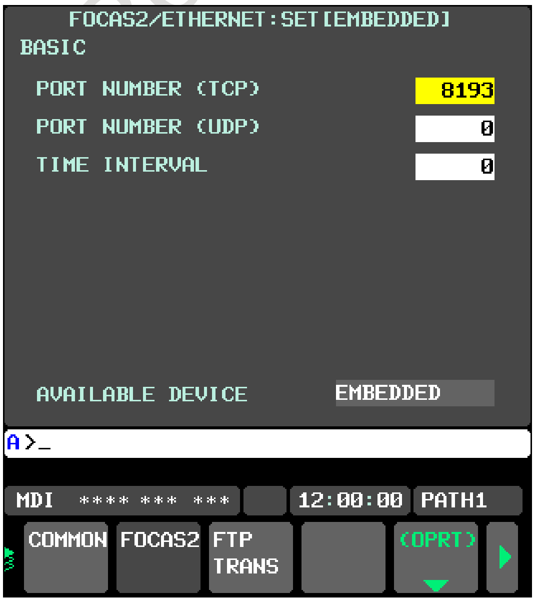


Fig. 17.2.6.2 (c) FOCAS2 screen

Setting items

Item	Description
PORT NUMBER (TCP)	Specify a port number to be used with the unsolicited messaging function (FOCAS2/Ethernet function). The valid input range is 5001 to 65535.
PORT NUMBER (UDP)	Set this item to 0.
TIME INTERVAL	Set this item to 0.

17.2.6.3 Mode selection

This subsection describes the selection of a mode for setting the unsolicited messaging function.

Unsolicited Message screen (BASIC)

Press soft key [UNSOLI MSG]. The Unsolicited Message screen (BASIC) (Fig. 17.2.6.3(a)) is displayed.

Fig. 17.2.6.3 (a) Unsolicited Message screen 1 (BASIC)

Setting items

Item	Description
MODE	Select a mode for setting the unsolicited messaging function. For the method of selection, see "**Operation**" described later. - When "CNC MODE" is selected This mode enables setting on the CNC screen. In this case, setting on the personal computer is disabled. For details, see Subsection 18.2.4.4, "Setting on the CNC screen". - When "PC MODE" is selected This mode enables setting on the personal computer. In this case, setting on the CNC screen is disabled. For details, see Subsection 18.2.4.5, "Setting on the personal computer".

NOTE

1. The mode is set to "PC MODE" at the time of initial use.
2. The mode can be switched only in the "Not Ready" state. For the "Not Ready" state, see "**Display items**" provided later.
3. If the mode is switched from "CNC MODE" to "PC MODE", all parameters set on the CNC screen are cleared.
4. The mode can be switched on the available device side only.

Operation

The mode can be switched as described below.

1 Press soft key [(OPRT)]. Soft key [MODE] is displayed. (Fig. 17.2.6.3(b))

Fig. 17.2.6.3 (b)

2 Press soft key [MODE]. Soft keys [CNC MODE] and [PC MODE] are displayed. (Fig. 17.2.6.3(c))

Fig. 17.2.6.3 (c)

Display items

Item	Description
IP ADDRESS	Displays the IP address of the personal computer currently connected. (Example of display format: "192.168.0.1")
STATUS	Displays the current state. The following five states are available: <1> Not Ready State where data is not transmitted even when a request for data transmission is made from an NC program or ladder program <2> Ready State where data is transmitted when a request for data transmission is made from an NC program or ladder program <3> Sending... State present from the acceptance of a request for data transmission from an NC program or ladder program until data transmission is completed <4> Receiving... State present from completion of data transmission until response data is received <5> Completed State present from reception of response data until response data processing is completed [Supplement] - Data transmission Means unsolicited message transmission (CNC→PC). - Response data Means a response to an unsolicited message (PC→CNC).
AVAILABLE DEVICE	Device where embedded Ethernet is currently enabled. The embedded Ethernet port or PCMCIA Ethernet card is displayed.

NOTE

1. To switch the state from "Not Ready" to "Ready", the FOCAS2 function cnc_unsolicstart needs to be executed on the personal computer.
2. To switch the state from other than "Not Ready" to "Not Ready", the FOCAS2 function cnc_unsolicstop needs to be executed on the personal computer.
3. For the timing charts of the states, see Subsection 18.2.6.6, "Execution methods".

17.2.6.4 Setting on the CNC screen

This subsection describes the method of setting on the Unsolicited Message screen.

NOTE

1 To enable the settings on the CNC screen and perform unsolicited messaging, the procedure below needs to be used.
   (1) Set all setting items on the Unsolicited Message screen (CONNECT).
   (2) Press soft key [(OPRT)] then press soft key [APPLY].
   (3) Start unsolicited messaging (execute the FOCAS2 function cnc_unsolicstart) on the personal computer.

2 Setting of the setting items on the Unsolicited Message screen (CONNECT) and execution of the soft keys ([(OPRT)] then [APPLY]) are possible only in the "Not Ready" state. For the "Not Ready" state, see "**Display items**" in Subsection 18.2.6.3, "Mode selection".

3 On the setting screen on the unavailable device side, the setting items can be set. However, execution of the soft keys ([(OPRT)] then [APPLY]) is possible on the available device side only.

Unsolicited Message screen (CONNECT)

Press soft key [UNSOLI MSG] then open page 2 and page 3 with page keys PAGE PAGE. The Unsolicited Message screen (CONNECT) (Figs. 17.2.6.4(a) and (b)) is displayed.

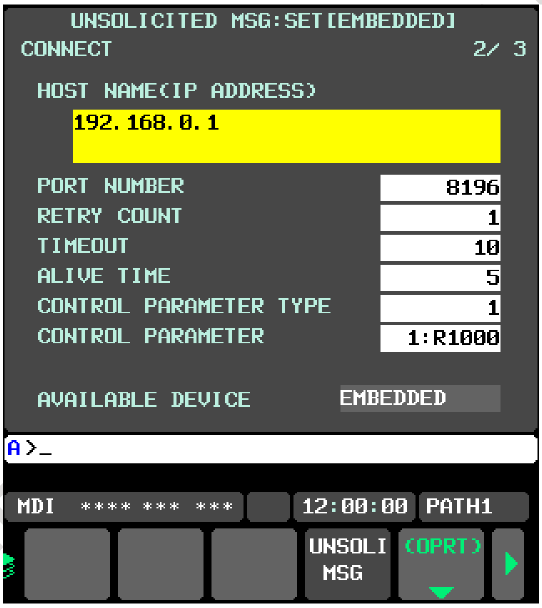


Fig. 17.2.6.4 (a) Unsolicited Message screen 2 (CONNECT)

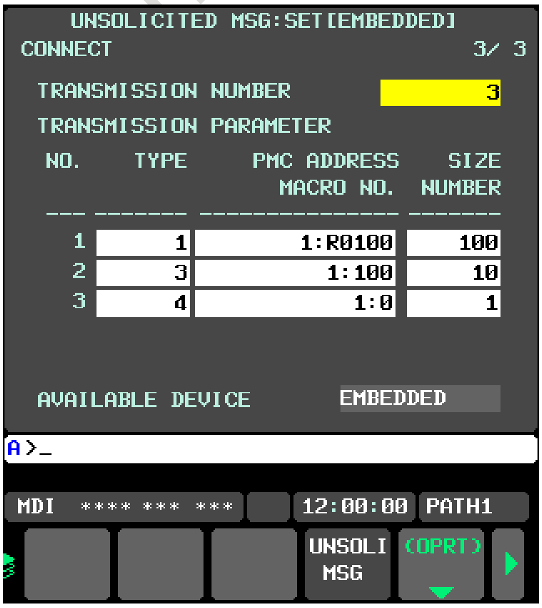


Fig. 17.2.6.4 (b) Unsolicited Message screen 3 (CONNECT)

Setting items

Item	Description
HOST NAME (IP ADDRESS)	When the DNS client function is disabled, specify the IP address of the communication destination personal computer. (Example of specification format: "192.168.0.1") When the DNS client function is enabled, specify the host name of the communication destination personal computer. (Up to 63 characters can be specified.) (Example of specification format: "UNSOLI-SRV.FACTORY")
PORT NUMBER	Specify the TCP port number and UDP port number of the communication destination personal computer. Usually, specify "8196". The valid input range is 5001 to 65535.
RETRY COUNT	Specify the number of retries to be made when there is no response to data transmitted by the communication function. The valid input range is 0 to 32767.
TIMEOUT	Specify a time-out period (in sec) from the transmission of data by the communication function until a response is made to the transmitted data. The valid input range is 1 to 32767.
ALIVE TIME	Specify the time interval (in sec) of the alive signal to be transmitted while the communication function is operating normally. Specify a value not greater than the value of TIMEOUT. The valid input range is 1 to 32767.
CONTROL PARAMETER TYPE	Specify a type of control parameter. When this parameter is set to 0, the control parameter is invalid. - When set to 1: PMC address (response notification method) - When set to 2: PMC address (simplified method) - When set to 3: Custom macro variable (simplified method) - When set to 4: Volatile RTM variable (simplified method) **(Note)** The RTM variable is a real-time custom macro variable.
CONTROL PARAMETER	Specify a control parameter for executing data transmission. When CONTROL PARAMETER TYPE is set to 1 Specify a PMC address for control. A PMC address in the R area or E area may be specified. Two bytes starting at a specified address are allocated in the area. When CONTROL PARAMETER TYPE is set to 2 Specify a PMC address for control. A PMC address in the R area or E area may be specified. Only a specified address (one byte) is allocated in the area. When CONTROL PARAMETER TYPE is set to 3 Specify a custom macro variable number for control. Only a volatile common variable may be specified as a custom macro variable. Only the variable with a specified variable number is allocated in the area. When CONTROL PARAMETER TYPE is set to 4 Specify a RTM variable number for control. Only a volatile RTM variable may be specified as a RTM variable. Only the variable with a specified variable number is allocated in the area.
TRANSMISSION NUMBER	Specify the number of data items to be transmitted. The valid input range is 1 to 3.

Item		Description	
TRANSMISSION PARAMETER (No.1 to 3)		Specify each parameter for transmission data.	
	TYPE	Specify a transmission data type. When this parameter is set to 0, the transmission parameter is invalid. - When set to 1 or 2: PMC address - When set to 3: Custom macro variable - When set to 4: Volatile RTM variable - When set to 5: Nonvolatile RTM variable	
	PMC ADDRESS or MACRO NO.	Specify the start of a transmission data area	
		When TYPE is set to 1 or 2	
			Specify a PMC address for transmission.
		When TYPE is set to 3	
			Specify a custom macro variable number for transmission.
		When TYPE is set to 4 or 5	
			Specify a RTM variable number for transmission.
	SIZE or NUMBER	Specify the size of a transmission data area or the number of variables. The maximum specifiable number of bytes is as follows: - When TRANSMISSION NUMBER is set to 1: 2890 bytes - When TRANSMISSION NUMBER is set to 2: 2874 bytes in total - When TRANSMISSION NUMBER is set to 3: 2858 bytes in total When using macro variables (custom macro variables or RTM variables), use a conversion rate of one variable for eight bytes.	
		When TYPE is set to 1 or 2	
			Specify a PMC area size (bytes) for transmission.
		When TYPE is set to 3	
			Specify the number of custom macro variables for transmission. When a macro variable number of 1000 or greater (system variable) is used, this parameter can be set to 1 only.
		When TYPE is set to 4 or 5	
			Specify the number of RTM variables for transmission.

⚠ **WARNING**

When setting PMC address or macro variable for the unsolicited messaging function, thoroughly confirm the content of “GENERAL WARNINGS FOR CNC APPLICATION DEVELOPMENT” in “SAFETY PRECAUTIONS” at the beginning of this manual.
Ensure that a PMC address area for control never overlaps PMC areas used for a purpose other than the unsolicited messaging function and that a macro variable for control never be doubly specified as a variable to be used for a purpose other than the unsolicited messaging function.
If the machine is operated without confirmation of above explanation, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

> ⚠ **CAUTION**
> 1 Immediately after the power is turned on, all R area and E area as volatile memory in the PMC area are 0.
> 2 The E area in the PMC area is normally allocated to volatile memory. However, it can also be used as nonvolatile memory.
> When the area is used as nonvolatile memory, the contents of the area are retained even after the power is turned off. So, special attention should be paid not to cause an unpredictable operation when the power is turned on next time.

> **NOTE**
> 1 Two methods are available for PMC address specification in CONTROL PARAMETER TYPE: response notification method and simplified method. For details of the methods, see Subsection 18.2.6.6, "Execution methods".
> 2 The valid setting range of PMC addresses depends on the usable PMC memory type. For details, refer to "PMC PROGRAMMING MANUAL (B-64513EN)".
> 3 The valid setting ranges of custom macro variable numbers and RTM variable numbers depend on the selected options. For details, refer to "OPERATOR'S MANUAL (COMMON TO LATHE SYSTEM/MACHINING CENTER SYSTEM) (B-64694EN)".
> 4 Ensure that the setting of TRANSMISSION NUMBER matches the settings of TRANSMISSION PARAMETER (No. 1 to No. 3). If TRANSMISSION NUMBER is set to 3, and an invalid value is specified in any of TRANSMISSION PARAMETER No. 1 to No. 3, for example, execution of soft key [APPLY] results in an error.

Setting PMC address

To setting a PMC address for control or a PMC address for transmission, specify it as follows:

Input format) <Path number>:<PMC address>

For example, for R0500 on the second path of the PMC, input "2:R500".
If <Path number> is omitted (R500), the first path is assumed (1:R0500).
If the <:> key is not available, it can be substituted with the </> key or the <EOB> key. ":" is optional.
To clear "<Path number>:<PMC address>" previously set, input " " (blank). ("---" will be displayed). In this case, it is assumed that no PMC area is used.

Setting macro variable

To setting a macro variable for control or a macro variable for transmission, specify it as follows:

Input format) <Path number>:<Variable No>

For example, for No.100 on the second path of the Macro, input "2:100".
If <Path number> is omitted (100), the first path is assumed (1:100).
If the <:> key is not available, it can be substituted with the </> key or the <EOB> key. ":" is optional.
To clear "<Path number>:<Variable No>" previously set, input " " (blank). ("---" will be displayed). In this case, it is assumed that no macro variable is used.

Operation

The settings of all setting items on the Unsolicited Message screen (CONNECT) can be made effective as follows:

1 Press soft key [(OPRT)]. Soft key [APPLY] is displayed. (Fig. 17.2.6.4(c))

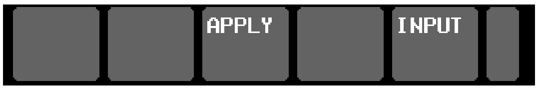


Fig. 17.2.6.4 (c)

2 Press soft key [APPLY].

17.2.6.5 Setting on the personal computer

For setting on the personal computer, create and set an application by using the following FOCAS2 functions:

- cnc_wrunsolicprm2 Parameter setting 2 for unsolicited messaging
- cnc_unsolicstart Start of unsolicited messaging

For details, refer to Chapter 5, "UNSOLICITED MESSAGING FUNCTION" in "Data Window Library Specifications" that comes with the FOCAS1/2 library.

> ⚠ **WARNING**
> When setting PMC address or macro variable for the unsolicited messaging function, thoroughly confirm the content of “GENERAL WARNINGS FOR CNC APPLICATION DEVELOPMENT” in “SAFETY PRECAUTIONS” at the beginning of this manual.
> Ensure that a PMC address area for control never overlaps PMC areas used for a purpose other than the unsolicited messaging function and that a macro variable for control never be doubly specified as a variable to be used for a purpose other than the unsolicited messaging function.
> If the machine is operated without confirmation of above explanation, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

> **NOTE**
> 1 To start unsolicited messaging, the FOCAS2 function cnc_wrunsolicprm2 needs to be executed first then the FOCAS2 function cnc_unsolicstart needs to be executed.
> 2 The FOCAS2 function cnc_wrunsolicprm2 can be executed only in the "Not Ready" state. For details of the state, see "**Display items**" in Subsection 17.2.6.3, "Mode selection".
> 3 When the FOCAS2 function cnc_wrunsolicprm2 is executed, the Unsolicited Message screen (CONNECT) displays the settings made on the personal computer.

17.2.6.6 Execution methods

How to execute the unsolicited messaging function is described below.
To execute the unsolicited messaging function, three methods are available:

- Using a PMC address for control based on the response notification method in a ladder program
- Using a PMC address for control based on the simplified method in a ladder program
- Using a macro variable for control based on the simplified method in an NC program

> **NOTE**
> 1 When a ladder program is used, the response notification method and the simplified method are available. A major difference is that the response notification method sends RES_CODE to the ladder program in response to data transmission but the simplified method does not send a response.
> To utilize a ladder program based on logic used with the Series 16*i*, for example, use the response notification method. When RES_CODE is unnecessary or a new ladder program is created, the simplified method can be used.
> 2 RES_CODE is recorded on the Ethernet log screen when a value other than 0x00 and 0x01 is detected.
> 3 For details of RES_CODE, refer to Chapter 5, "UNSOLICITED MESSAGING FUNCTION" in "Data Window Library Specifications" that comes with the FOCAS1/2 library.

When a PMC address for control is used (response notification method)

A description of using a PMC address for control in a ladder program according to the response notification method is provided below.

> **NOTE**
> A combination of a PMC address for control and a macro variable for transmission is also usable. In this case, note that the read timing of the value of a macro variable to be transmitted cannot be identified when viewed from the ladder program.
> In the description below, a PMC address is used for both of control and transmission.

Explanation of PMC address signals for control

A detailed description of PMC address signals for control used to execute the unsolicited messaging function is provided below. A PMC address area for control consists of 2 bytes.
The description below assumes that Rxxxx (with no PMC path number specified) is used as a PMC address for control.

No.	#7	#6	#5	#4	#3	#2	#1	#0
Rxxxx	REQ							

REQ <Rxxxx.7>

[Name] Message transmission request signal
[Classification] Input signal
[Function] Requests transmission of an unsolicited message.
[Operation] After preparing a transmission message at a PMC address for transmission, the ladder program sets this signal to “1”. The message is then transmitted to the personal computer.

No.	#7	#6	#5	#4	#3	#2	#1	#0
Rxxxx+1	RES	COM	RES_CODE					

RES <Rxxxx+1.7>
[Name] Message response reception signal
[Classification] Output signal
[Function] Posts the reception of a response to an unsolicited message.
[Output condition] Upon reception of a message by the personal computer, a response to the message is transmitted to the CNC (communication function). When the CNC (communication function) receives the response, this signal is set to “1”. When this signal is set to “1”, the ladder program reads RES_CODE then clears REQ to “0”. Next, the CNC (communication function) clears RES_CODE to “0” then sets this signal to “0”.

COM <Rxxxx+1.6>
[Name] Message transmission start signal
[Classification] Output signal
[Function] Posts the start of transmission of an unsolicited message.
[Output condition] When transmission of a message to the personal computer is started, this signal is set to “1”. Upon completion of message transmission, this signal is set to “0”.

RES_CODE <Rxxxx+1.0> to <Rxxxx+1.5>
[Name] Message response reception result signal
[Classification] Output signal
[Function] Posts the reception result of a response to an unsolicited message.
[Output condition] The reception result of a response to a message is set. After reading this signal, the ladder program clears REQ to “0”. The CNC (communication function) then clears this signal to “0”.

> **NOTE**
> For details of RES_CODE, refer to Chapter 5, "UNSOLICITED MESSAGING FUNCTION" in "Data Window Library Specifications" that comes with the FOCAS1/2 library.

Timing chart of PMC address signals for control

The timing chart of PMC address signals for control based on the response notification method is described in the Fig. 17.2.6.6 (a).
In the example below, an unsolicited message is transmitted once after reception of the FOCAS2 function cnc_unsolicstart then the FOCAS2 function cnc_unsolicstop is received.

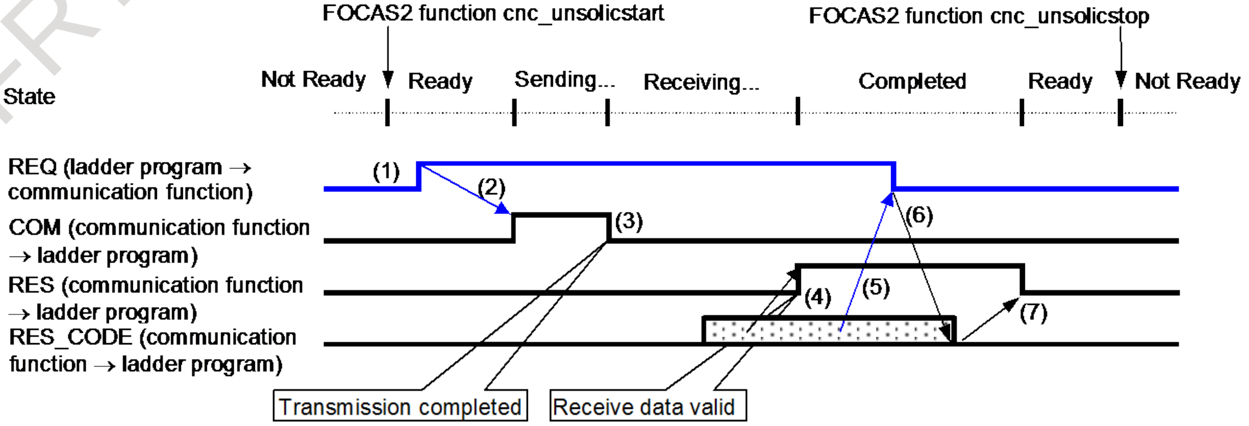


Fig. 17.2.6.6 (a)

(1) After checking that RES is set to “0”, the ladder program prepares a message then sets REQ to “1”.
(2) Because of REQ set to “1”, the communication function sets COM to “1” then transmits the message.
(3) Upon completion of message transmission, the communication function sets COM to “0”.
(4) Upon reception of a response to the message, the communication function sets RES_CODE then sets RES to “1”.
(5) Because of RES set to “1”, the ladder program reads RES_CODE then sets REQ to “0”.
(6) Because of REQ set to “0”, the communication function clears RES_CODE to “0”.
(7) The communication function sets RES to “0”.

NOTE
For details of the states, see "**Display items**" in Subsection 17.2.6.3, "Mode selection".

When a PMC address for control is used (simplified method)

A description of using a PMC address for control in a ladder program according to the simplified method is provided below.

NOTE
A combination of a PMC address for control and a macro variable for transmission is also usable. In this case, note that the read timing of the value of a macro variable to be transmitted cannot be identified when viewed from the ladder program.
In the description below, a PMC address is used for both of control and transmission.

Explanation of PMC address signals for control

A detailed description of PMC address signals for control used to execute the unsolicited messaging function is provided below. A PMC address area for control consists of 1 byte.
The description below assumes that Rxxxx (with no PMC path number specified) is used as a PMC address for control.

No.	#7	#6	#5	#4	#3	#2	#1	#0
Rxxxx	REQ							

REQ <Rxxxx.7>
[Name] Message transmission request signal
[Classification] Input/Output signal
[Function] Requests transmission of an unsolicited message.
[Operation] After preparing a transmission message at a PMC address for transmission, the ladder program sets this signal to 1. The message is then transmitted to the personal computer. Upon reception of a response to the message, the CNC (communication function) clears this signal to 0.

Timing chart of PMC address signals for control

The timing chart of PMC address signals for control based on the simplified method is described below. In the example below (Fig. 17.2.6.6 (b)), an unsolicited message is transmitted once after reception of the FOCAS2 function cnc_unsolicstart then the FOCAS2 function cnc_unsolicstop is received.

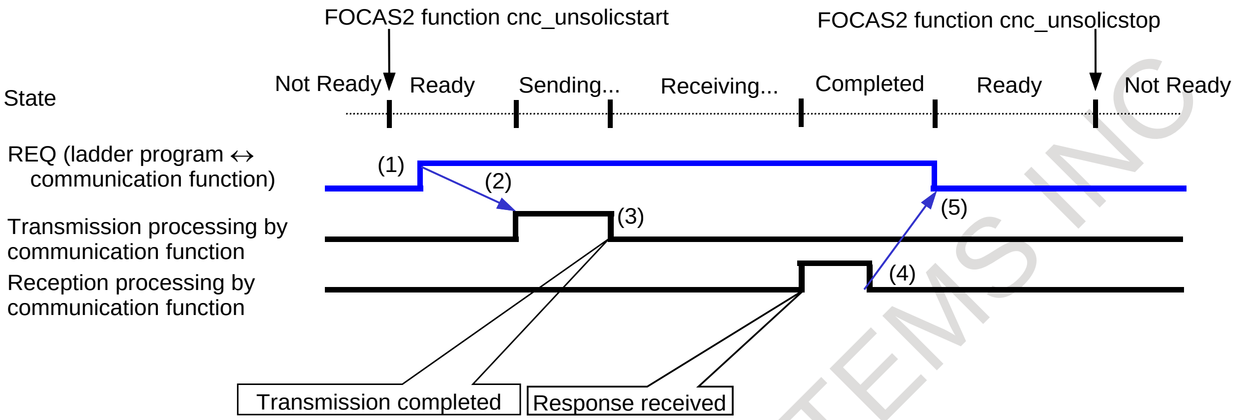


Fig. 17.2.6.6 (b)

(1) After checking that REQ is set to 0, the ladder program prepares a message then sets REQ to 1.
(2) Because of REQ set to 1, the communication function transmits the message.
(3) The communication function completes message transmission processing.
(4) Upon reception of a response to the message, the communication function completes reception processing.
(5) Because of reception processing completed, the communication function sets REQ to 0.

> **NOTE**
> For details of the states, see "**Display items**" in Subsection 17.2.6.3, "Mode selection".

When a macro variable for control is used (simplified method)

A description of using a macro variable for control in an NC program according to the simplified method is provided below.
Macro variables for control are classified as custom macros and RTM macros, but the same execution method is applicable.

> **NOTE**
> A combination of a macro variable for control and a PMC address for transmission is also usable. In this case, note that the read timing of the value of PMC data to be transmitted cannot be identified when viewed from the NC program.
> In the description below, a macro variable is used for both of control and transmission.

Explanation of a macro variable for control

A detailed description of a macro variable for control used to execute the unsolicited messaging function is provided below. One macro variable for control is used.
The description below assumes that #xxxx (with no CNC path number specified) is used as a macro variable number for control.

REQ <#xxxx>

[Name] Message transmission request signal
[Classification] Input/Output signal
[Function] Requests transmission of an unsolicited message.
[Operation] After preparing a transmission message in a macro variable for transmission, the NC program sets this signal to 1. The message is then transmitted to the personal computer. Upon reception of a response to the message, the CNC (communication function) clears this signal to 0.

> **NOTE**
> A REQ input/output value is a real number. So, "0" means "0.0", and "1" means "1.0".

Timing chart of a macro variable for control

The timing chart of a macro variable for control based on the simplified method is described below.
In the example below (Fig. 17.2.6.6 (c)), an unsolicited message is transmitted once after reception of the FOCAS2 function cnc_unsolicstart then the FOCAS2 function cnc_unsolicstop is received.

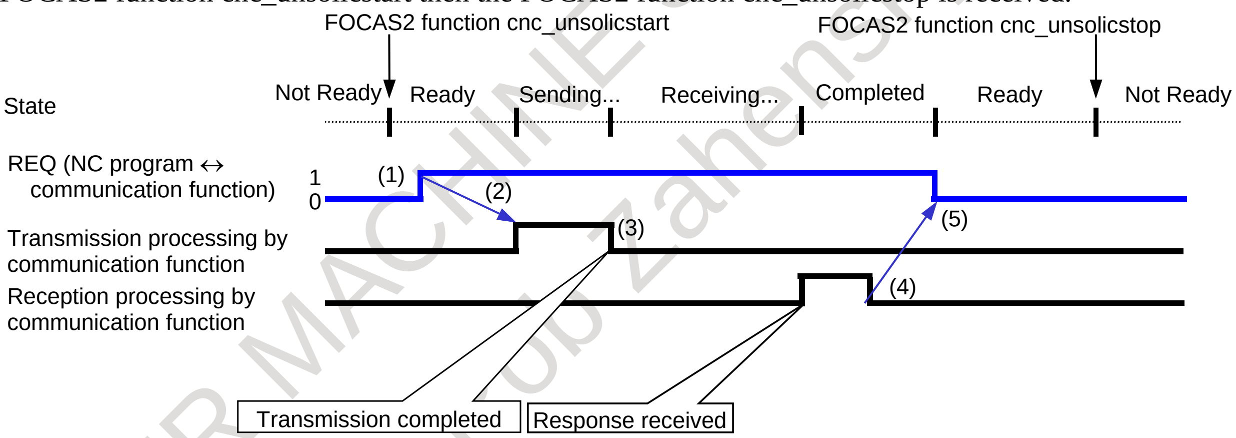


Fig. 17.2.6.6 (c)

(1) After checking that REQ is set to 0, the NC program prepares a message then sets REQ to 1.
(2) Because of REQ set to 1, the communication function transmits the message.
(3) The communication function completes message transmission processing.
(4) Upon reception of a response to the message, the communication function completes reception processing.
(5) Because of reception processing completed, the communication function sets REQ to 0.

> **NOTE**
> For details of the states, see "**Display items**" in Subsection 17.2.6.3, "Mode selection".

Example of using a macro variable for control

An example of using a macro variable for control is provided below.

- Example

[Description]

An NC program on CNC path number 1 posts NC command start date and time information as an unsolicited message to the personal computer.

[Setting]

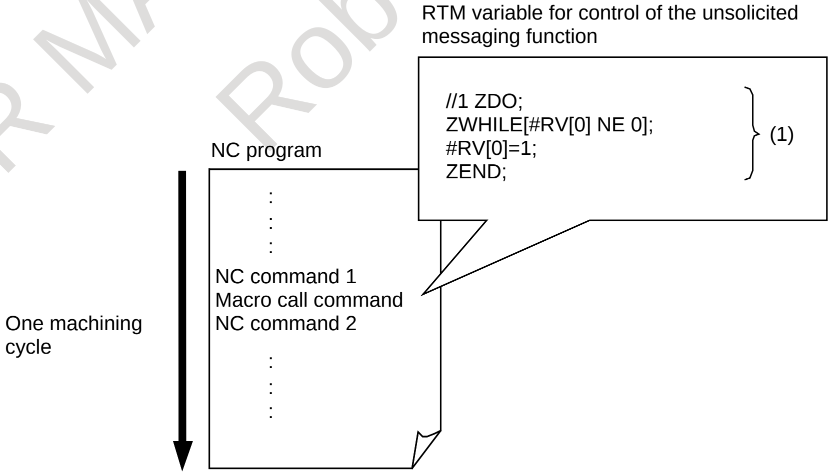
As a macro variable for control, volatile RTM variable number #0 (REQ) is used. On the other hand, macro variables for transmission are used for date and time information managed inside the CNC (system variable numbers #3011 and #3012).

Setting item		Set value
CONTROL PARAMETER TYPE		4
CONTROL PARAMETER		1:0
TRANSMISSION NUMBER		2
TRANSMISSION PARAMETER (NO.1)		
	TYPE	3
	MACRO NO.	1:3011
	NUMBER	1
TRANSMISSION PARAMETER (NO.2)		
	TYPE	3
	MACRO NO.	1:3012
	NUMBER	1

[Example of NC program]

A RTM variable is executed in synchronism with the immediately following NC command according to the RTM variable specification. At this time, the operation timing of the immediately following NC command is not affected.

So, the NC program indicated below posts NC command 2 start date and time information as an unsolicited message to the personal computer.



(Tip) (1) in the example above corresponds to the number in the timing chart provided earlier.

NOTE
1. Note that if a custom macro is used as a macro for control, the operation timing of an NC command is affected. If the RTM variable in the example above is replaced with a custom macro, for example, NC command 2 becomes unexecutable unless the macro variable for control is changed.
2. For details of custom macros and RTM variables, refer to "OPERATOR'S MANUAL (Common to Lathe System/Machining Center System) (B-64694EN)".

17.2.6.7 Related parameters

For embedded Ethernet port

	#7	#6	#5	#4	#3	#2	#1	#0
14880				UNM				

[Input type] Parameter input
[Data type] Bit

#4 UNM With an embedded Ethernet port, the CNC Unsolicited Messaging function is:
0: Not used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
14882				UNS				

[Input type] Parameter input
[Data type] Bit

#4 UNS In the CNC Unsolicited Messaging function of an embedded Ethernet port, when the end of the function is requested by other than the CNC Unsolicited Messaging server currently connected:
0: The request for the end of the function is rejected.
1: The request for the end of the function is accepted.

After these parameters are modified, the power must be turned off then back on or the embedded Ethernet function must be terminated then restarted for the settings to become effective.

17.2.7 Setting of the CNC Screen Web Server Function

This subsection describes the setting required to operate the CNC screen Web server function with the embedded Ethernet function.

17.2.7.1 Overview

This subsection provides an overview of the CNC screen Web server function and describes the execution procedure.

Overview of the CNC screen Web server function

An overview of the CNC screen Web server function is provided below.
This function enables screen display and screen switch of CNC on a Web browser of a Tablet-type device that connects with CNC through Ethernet.
The screen which is displayed on a Web browser is the same as the screen displayed on a standard CNC Display unit.
The screen switch operation at a Web browser is reflected in a standard CNC Display unit.

NOTE
1. A Tablet-type device cannot be used instead of a standard CNC Display unit.
2. It is also possible to use a Web browser of the personal computer.
3. Behavior might be different according to a Tablet-type device and a Web browser.
4. The update cycle of the screen on a Tablet-type device slows more than a standard CNC Display unit. In addition, it might become slower in the following cases.
   - System configuration with high CPU load, such as multi-path/multi-axes control.
   - Operation condition with high CPU load, such as program operation of small blocks.
   - Bad condition of communication between CNC and Tablet-type device.

⚠ **WARNING**
On the screen of Web browser, do not perform the operation which relates to the machine operation, e.g. Virtual MDI key function, the data input by soft key, or a machine operator's panel screen made by machine tool builder.
The bad condition of communication might cause an unexpected machine operation. For example, the off operation of a button on the screen might be lost, or the response of a button might be slow. Such an unexpected machine operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.
Therefore, when connecting with Web browser by this function, perform the sequences that check Web browser connection status signal WBCNT <F0578.2> and prohibit the operation such as the data input and the operation from the screen.

Connection status of the CNC screen Web server function

CNC connects a Tablet-type device via the embedded Ethernet and a wireless LAN equipment.
Refer to the manual of each equipment for the method of connecting the Tablet-type device and wireless LAN equipment.

By starting a Web browser on the Tablet-type device and then inputting a URI, the Tablet-type device is connected with CNC.

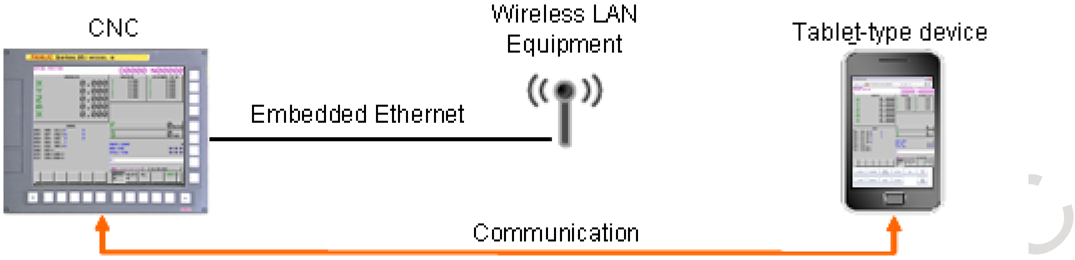


Fig.17.2.7.1 Connection

> **NOTE**
> 1 It is not possible to connect CNC with two or more Tablet-type devices at the same time. Only the first Tablet-type device can be connected.
> 2 It is not possible to connect CNC while the CNC screen is displayed on a personal computer by using CNC screen display function.

The display units of CNC that can be used are as follows.

Table17.2.7.1 (a) Display unit of CNC

CNC	Display unit
LCD Mounted Type	10.4" color LCD

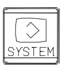
> **NOTE**
> 1 The update of CNC screen on a Tablet-type device might stop temporarily according to the CNC system configuration and the operation status. However, it doesn't influence automatic operation of CNC.
> 2 This function cannot be used in CNC that uses the following display.
> - Without standard CNC Display unit (Use PC connecting)
> - FANUC *i*HMI display unit (Use PANEL *i*H/*i*H Pro)
> - The secondary display unit for Ethernet connection
> - The shared display unit for Ethernet connection
> 3 This function cannot be used while connecting *i*Pendant.
> 4 This function cannot be used by connecting via a PCMCIA Ethernet card.

17.2.7.2 Setting of the WEB SERVER function

This subsection describes the setting of the WEB SERVER function for operating the CNC screen Web server function.

Procedure

1 Enable the CNC screen Web server function according to "Related parameters" described later.

2 Start the CNC again then press function key  .

3 Soft key [EMBED] is displayed. (Press the continuous menu key until the soft key is displayed.)

4 Press soft key [EMBED]. The Ethernet Setting screen for the embedded Ethernet port is displayed.

5 Press soft key [WEB]. The WEB SERVER screen is displayed. (Press the continuous menu key until the soft key [WEB] is displayed.)

6 After setting items, restart the embedded Ethernet function.

WEB SERVER screen

Press soft key [WEB]. The WEB SERVER screen (Fig. 17.2.7.2(a)) is displayed.

Fig.17.2.7.2 (a) WEB SERVER screen

Setting items

Item	Description
PORT NUMBER	Specify a port number to be used with the CNC screen Web server function. If 0 is specified, PORT NUMBER is treated as 80.
TIMEOUT	Specify a timeout of the CNC screen Web server function. TIMEOUT can be set the value of 0, 5 to 99 in seconds. If 0 is specified, TIMEOUT is treated as 10 seconds.
USER NAME	Specify a user name to be used for the CNC screen Web server function. (Up to 31 characters can be specified.)
PASSWORD	Specify a password for the user name specified above. (Up to 31 characters can be specified.) Be sure to set a password.

17.2.7.3 Operation of the CNC screen Web server function

This subsection describes the operation of the CNC screen Web server function.

Start of the CNC screen Web server function

Procedure

1. Turn on the power of the CNC.
2. Wait until the CNC screen is displayed.
3. Start a Web browser on the Tablet-type device and input URI (Uniform Resource Identifier) "http://<IP Address>(:[port number])/cncscrn/". The last "/" is not omissible.

   (Example 1) Case of IP Address (192.168.1.1) and PORT NUMBER (0 or 80)
   Input "http://192.168.1.1/cncscrn/" to the address bar.

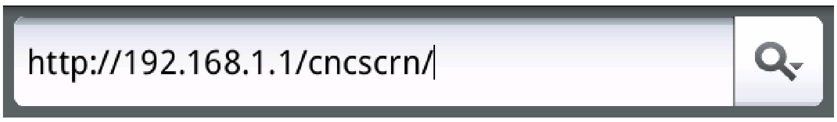


Fig.17.2.7.3 (a) Input case 1

(Example 2) Case of IP Address (192.168.1.1) and PORT NUMBER (8501)
Input "http://192.168.1.1:8501/cncscrn/" to the address bar.

Fig.17.2.7.3 (b) Input case 2

4 When the login screen is displayed, input the username and password, and push the login button.

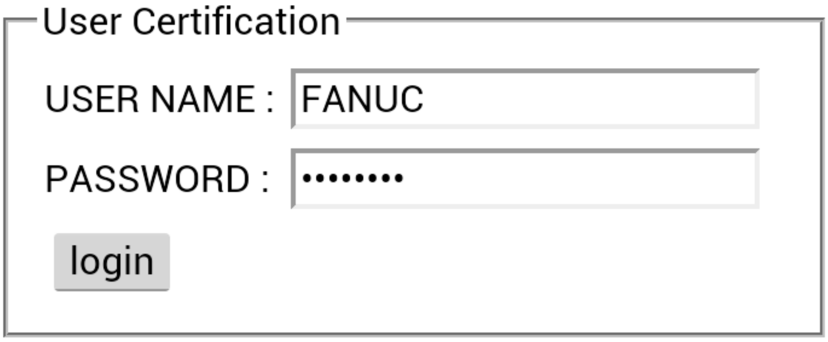


Fig.17.2.7.3 (c) login screen

> **NOTE**
> 1 When the Web browser connection prohibition signal WBEND <G0579.5> is set to "1", the error "Access Denied : Forbidden by signal" is displayed.
> 2 When the Web browser connection status signal WBCNT <F0578.2> is "1", the error "Access Denied : Doubly access" is displayed.
> 3 When bit 6 (WAL) of parameter No.14882 WAL (refer to chapter 4) is set to 0 and failing in login continuously five times, CNC screen Web server function becomes disabled until the CNC is restarted.

5 The CNC screen is displayed on a Web browser, and the Web browser connection status signal WBCNT <F0578.2> becomes “1”.

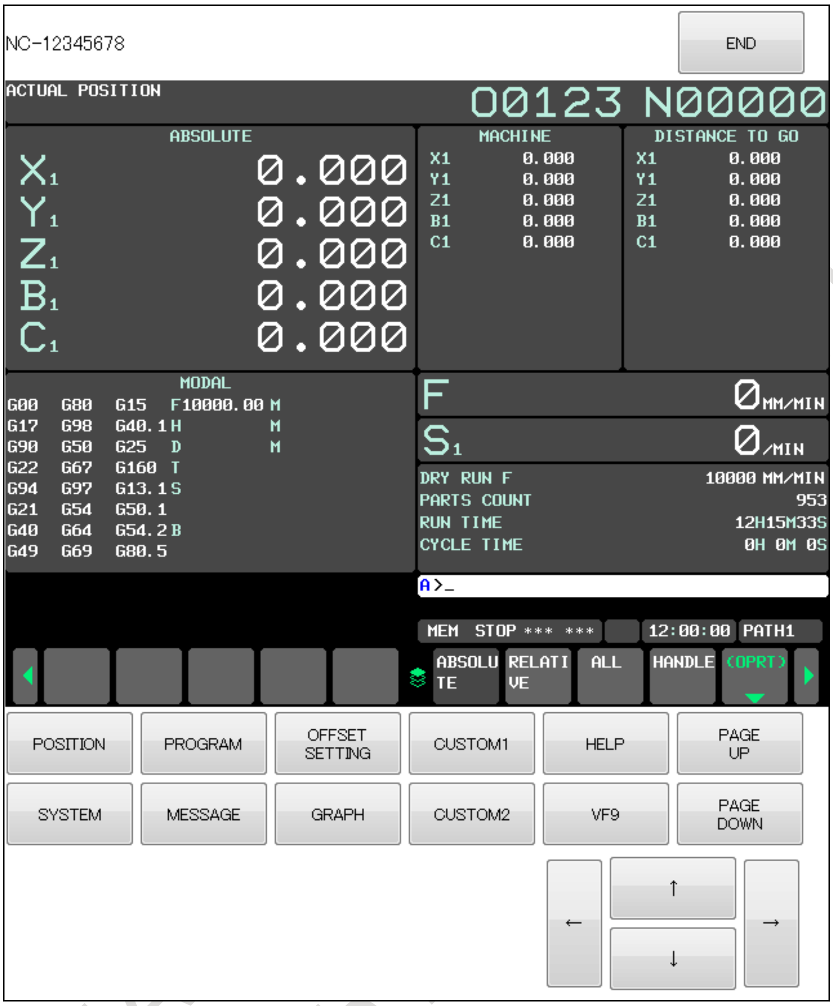


Fig.17.2.7.3 (d) CNC screen on a Web browser

> **NOTE**
> The update cycle of the screen slows more than a standard CNC Display unit.

End of the CNC screen Web server function

The communication with CNC ends in the following cases.

- When Web browser on Tablet-type device is ended.
- When the connection of CNC and Tablet-type device is cut.
- When the Web browser connection prohibition signal WBEND <G0579.5> is set to "1".
- When CNC screen function is started on other personal computer.
- When END button on the Web browser is pushed.

After the time interval that set on WEB SERVER screen is passed, the Web browser connection status signal WBCNT <F0578.2> becomes "0".
To reconnect with the CNC, execute procedure 3 of "Start of CNC screen Web server function" after the Web browser connection status signal WBCNT <F0578.2> becomes “0”.

> **NOTE**
> CNC cannot be operated with MDI key and touch panel of CNC while the Web browser connection status signal WBCNT <F0578.2> is “1”, even if the communication with CNC ends. The maximum time that CNC cannot be operated is the time having set to TIME INTERVAL.

Operation on Web browser

This subsection explains the operation method in a Web browser.

> **NOTE**
> While connecting with Web browser, neither MDI key nor touch panel of CNC can be used for operation.
> The reset key of CNC and External key input, however, are always usable on the CNC.

Operation by button

Function keys, help key, and page keys are displayed as buttons.
CNC screen changes according to each key if these buttons are pushed.
Selected item on the screen is changed if cursor keys are pushed.
If [VF9] button is pushed, short-cut menu by vertical keys is displayed and the operation by touch screen becomes possible.
[End] button is displayed in the upper part of screen.
The communication with CNC ends by pushing the [END] button.

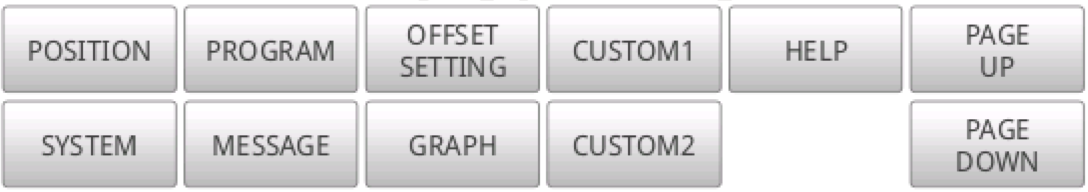


Fig.17.2.7.3 (e) Button on a Web browser

Operation by touch screen

Tap	Soft key on the CNC screen and button can be pushed. CNC can be operated with each key by pushing function key and page key.
Double tap	CNC screen size is set to be suitable for the screen size of Tablet-type device.
Pinch out / Pinch in	CNC screen is expanded and reduced. Pinch out is expansion, and Pinch in is reduction.

Refer to the manual of Tablet-type device for explanation of each operation.

> **NOTE**
> 1 To perform the Tap operation on the screen made by machine tool builder using C language executor, it is necessary to realize the operation with touch panel.
> 2 The communication with CNC might become interrupted during the operation of Pinch out / Pinch in. If the time having set to TIME INTERVAL passed, the communication of CNC and a Tablet-type device is cut.

Operation by direction of Tablet-type device

The direction of screen is changed automatically according to the direction of Tablet-type device.

Operation by key input

When the value of parameter KIE is one, the text input area, SEND button and CLEAR button next to the cursor keys, and the key input becomes possible.

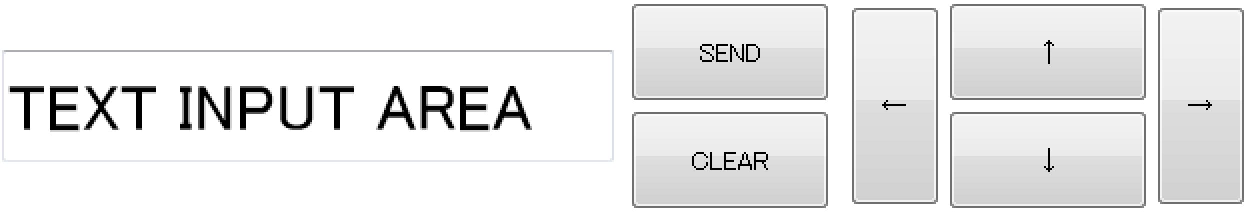


Fig.17.2.7.3 (f) Button on a Web browser

If SEND button is pushed, the character string in the text input area is transmitted to the key-in buffer of CNC. If CLEAR button is pushed, the key-in buffer of CNC is cleared.
The following characters can be input to the text input area.

Enable character	Remarks
a-z, A-Z, 0-9	Alphanumeric characters
!”#&’()*+,-<=>?@[/]^`~.;:~	Special symbols
Space	Blank

NOTE

“{“, ”|”, ”}”, “$”, “¥” and 2-byte code used for Japanese cannot be sent to the key-in buffer of CNC.

Input / Output of CNC data

CNC data is input/output from/to CNC according to the setting of CNC by operation on Web browser screen.
The data cannot be input/output from/to the Web browser side.

Display language

The title and buttons on Web browser are displayed in English.

NOTE

The display language of CNC screen is displayed according to the setting of CNC.

Type of display screen

The screens of CNC displayed in Web browser is the same as the screens displayed in CNC. It is not possible to display different size or kind of screens from CNC.

NOTE

1 Boot screen and IPL screen cannot be displayed.
2 When the system alarm is generated by CNC, the connection of Tablet-type device is cut. The system alarm screen is not displayed in Web browser.

17.2.7.4 Related signals

Signal

Web browser connection prohibition signal WBEND <G579.5>

[Classification] Input signal
[Function] Forbid the connection with Web browser.
[Operation] "0": Permit the connection with Web browser.
"1": Forbid the connection with Web browser.
The connection is shut while connecting with Web browser.

Web browser connection status signal WBCNT<F578.2>

[Classification] Output signal
[Function] Show the status of the connection with Web browser.
[Operation] This signal is set to 1 when connecting with Web browser.
This signal is set to 0 when not connecting with Web browser.

NOTE
Perform the sequences that check this signal and prohibit the operation which relates to the machine operation such as the data input and the operation from the screen, e.g. Virtual MDI key function, the data input by soft keys, or the operation on a machine operator's panel screen made by machine tool builder.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0579			WBEND					

	#7	#6	#5	#4	#3	#2	#1	#0
F0578						WBCNT		

17.2.7.5 Related parameters

	#7	#6	#5	#4	#3	#2	#1	#0
11373	WSE							

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#7 WSE CNC screen Web server function is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
14882		WAL	KIE					

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#5 KIE In CNC screen Web server function, key input is:
0: Disabled.
1: Enabled.

#6 WAL In CNC screen Web server function, when failing in login continuously five times, this function becomes:
0: Disabled.
1: Enabled.

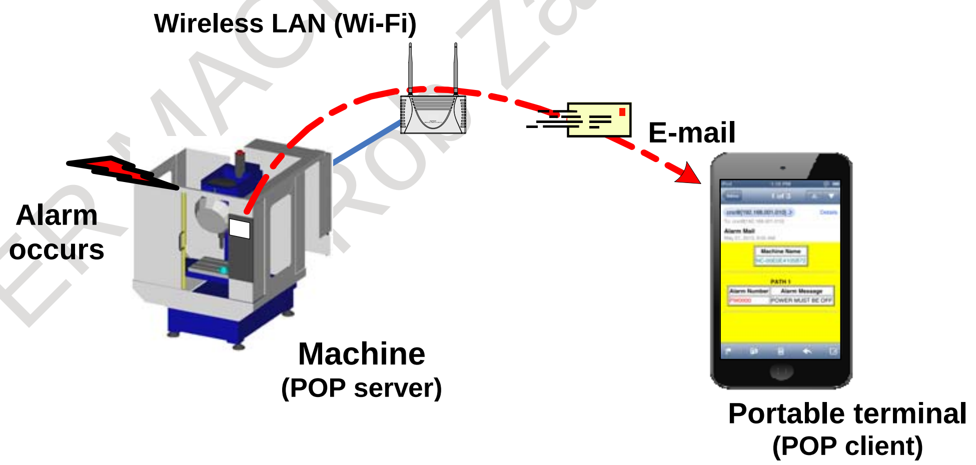
> **NOTE**
> When CNC screen Web server function has become disabled, restart the CNC.

17.2.8 CNC Status Notification Function

This subsection describes the setting required to operate the CNC status notification function with the embedded Ethernet function.

17.2.8.1 Overview

CNC status notification function is provided in the method of Monitoring the alarm even if the operator is not in front of the machine.
This function delivers E-mail of the information about CNC status (for example, alarm number and alarm message) to the e-mail software of a portable terminal or a personal computer via the Embedded Ethernet..



NOTE

1. This function can be used on Embedded Ethernet port and cannot be used on PCMCIA LAN card.
2. This function does not use an e-mail server on the way, to deliver E-mail. Therefore, this function is used only in the Intranet. E-mail cannot be delivered to an external device on Internet.
3. The e-mail software of a portable terminal reads E-mail from CNC with this function. Therefore, E-mail is read according to the timing that the e-mail software of a portable terminal acquires.
4. In the above figure, CNC communicates with a portable terminal by using wireless LAN. However, it is also possible to communicate with a personal computer by using wired LAN.
5. This function is not "short message service(SMS)" using the cellular phone network.
   Therefore, the cellular phone not to be connectable with wireless LAN cannot be used.

17.2.8.2 Contents of E-mail

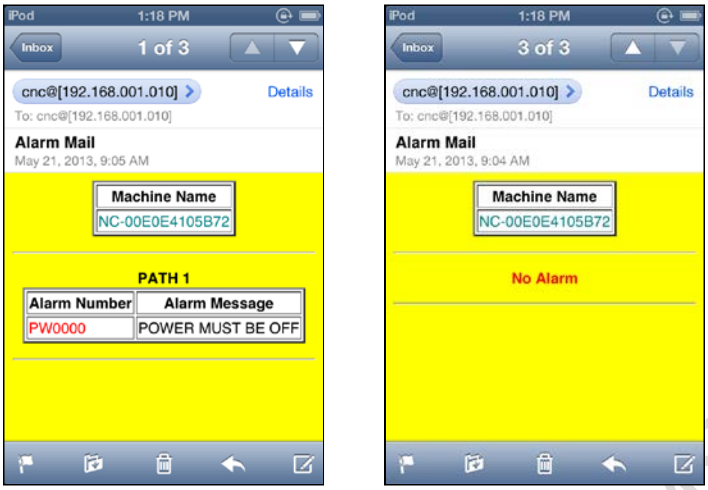
This function creates E-mail and delivers it to the mail software of a portable terminal and a personal computer.
There are two kinds of E-mail, "Alarm mail" and "Status mail".
When the displayed language of the CNC screen is Japanese, English, or Simplified Chinese, the title and each item name of the mail can be displayed in each language. In the case of other languages, the title and each item name of the mail are displayed in English.

Alarm Mail

The alarm mail reports the information of CNC alarm status.

The alarm mail is made at the following cases.
(1) When the alarm is newly generated
(2) When the alarm is canceled
(3) When the number of generated alarms is changed

When two or more alarms have been generated, those alarms are constructed into one alarm mail.
The number of alarms included in one alarm mail is up to 10, and more alarms are not reported.
The order of the notified alarm is the order displayed on the alarm screen of CNC.
In the multi-path system, the alarms are gathered in order of path. Therefore, for example if ten alarms or more are generated in the first path, the alarms which are generated in the second path or later are not reported.
Moreover, the alarm message is reported when the displayed language of the CNC screen is Japanese, English, and Simplified Chinese.
Only the alarm number is written in the case of other languages.



When the alarm is generated **When the alarm is canceled**

Fig.17.2.8.2 (a) Alarm Mail

> **NOTE**
> When the following alarms are generated, the alarm status is reported.
> - PS/BG/SR/SW/SV/OT/IO/PW/SP/OH/DS/IE alarms
> - External alarm message (EX alarm)
> - Macro alarm message of #3000 (MC alarm)

Status Mail

The status mail reports the counter values of the total number of machined parts, number of machined parts and number of parts required when bit 5 (SNP) of parameter No. 14884 is one.

The status mail is made at the following cases:
(1) When the counter values of number of machined parts is changed.

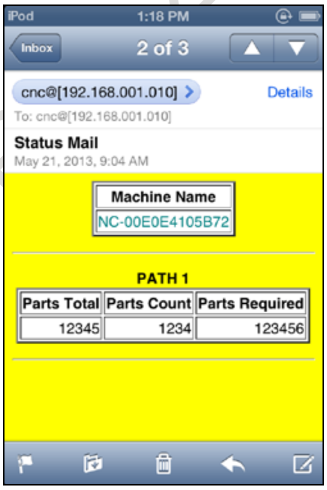


Fig.17.2.8.2 (b) Status mail

> **NOTE**
> The total number of machined parts, number of machined parts and number of parts required of only the first path are reported even in the case of multi-path CNC.
> Even if the number of machined parts of the second CNC path or later is changed, it is not reported.

Message Mail

The message mail reports the message (external operator message or macro message) required when parameter SNO (No.14884#4) is one.

The message mail is made at the following cases:

(1) When the message is newly generated
(2) When the number of generated message is changed
(3) When the message is canceled

When two or more messages have been generated, those messages are constructed into one message mail.
The number of messages included in one message mail is up to 3, and more messages are not reported.
The order of the notified message is the order displayed on the operator message screen of CNC.
The message number, message content, and machine group number of each machine group are notified in external operator message.
The message and path number of each path are notified in macro message.
In the multi-path system, the messages are gathered in order of the path regardless of machine group. And in the path, the messages are gathered in order of external operator message and macro message. Therefore, for example in the machine configuration to which path 1 and 2 belong to machine group 1 and path 3 belong to machine group 2, if three external operator messages or more are generated in the machine group 1, the macro message generated in the machine group 1 and the message generated in the machine group 2 are not reported.
And if one external operator message is generated in the machine group 1, one macro message is generated in the path 1 and one external operator message is generated in the machine group 2, the macro message generated in the path 3 is not reported.
Moreover, the message content is reported when the displayed language of the CNC screen is Japanese, English, and Simplified Chinese.
Only the message number (or the string "MCR" in Macro message) is reported in the case of other languages.

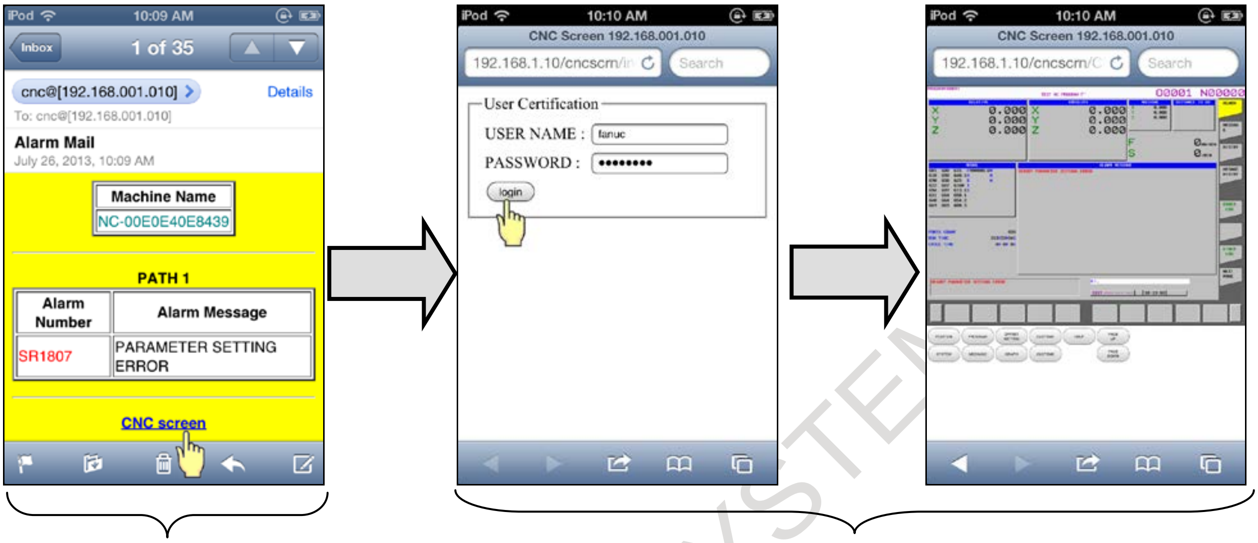
When the message is generated **When the message is canceled**

Fig.17.2.8.2 (c) Message mail

> **NOTE**
> As for the macro message, this function supports English only.

17.2.8.3 Cooperation with CNC screen Web server function

URI(CNC screen) of CNC screen Web server function is added in the mail when CNC screen Web server function is enabled. The CNC screen can be displayed on Web a browser by clicking



Mail client Web browser

Fig.17.2.8.3 Cooperation with CNC screen Web server function

> **NOTE**
> Please refer to Technical Report of CNC Screen Web server function for details of this function.

17.2.8.4 Setting of CNC Status Notification function

This section describes how to set the CNC parameters for CNC Status Notification function.
The setting of the Portable terminal (POP client), please refer to the manual of the Portable terminal or Technical Report (A-94974)

Procedure

(1) Press function key SYSTEM.

(2) Soft key [EMBED PORT] appear.
(When there is no soft keys, press the continue key.)

(3) Press soft key [EMBED PORT] soft key, the Ethernet Setting screen for embedded Ethernet port is displayed.

(4) Press soft key [COMMON] and [MAIL] and then enter parameters for the items that appear.

COMMON screen (BASIC)

Press soft key [COMMON]. The COMMON screen (BASIC) is displayed.

Fig. 17.2.8.4 (a) COMMON screen (BASIC)

Setting items

Item	Description
IP ADDRESS	Specify the IP address of the embedded Ethernet. (Example of specification format: "192.168.0.100")
SUBNET MASK	Specify a mask address for the IP addresses of the network. (Example of specification format: "255.255.255.0")
ROUTER IP ADDRESS	Specify the IP address of the router. Specify this item when the network contains a router. (Example of specification format: "192.168.0.253")
DHCP CLIENT	The value of parameter No.14880#6(DHC) related to the DHCP client function is displayed, and the setting is also possible. For details, see Subsection, "Setting Up the DNS/DHCP Function".

Display items

Item	Description
MAC ADDRESS	Embedded Ethernet MAC address
AVAILABLE DEVICE	Enabled device of the embedded Ethernet. Either the embedded Ethernet port or the PCMCIA Ethernet card is displayed.

COMMON screen (DETAIL)

After pressing soft key [COMMON], press either page key PAGE↑ PAGE↓ to call a desired COMMON screen (DETAIL).

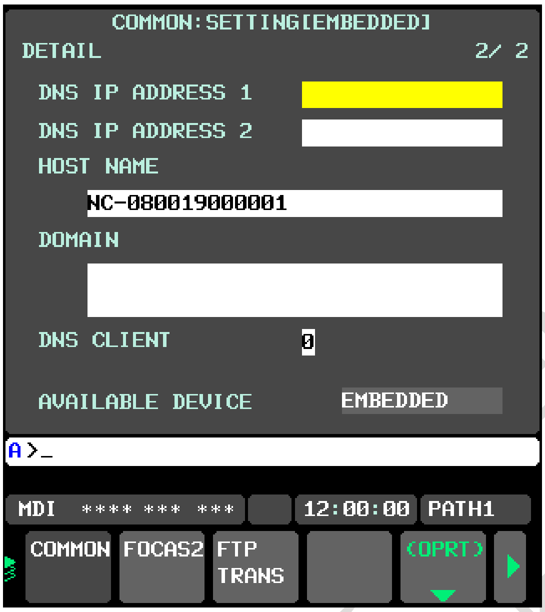


Fig. 17.2.8.4 (b) COMMON screen (DETAIL)

Setting items

Item	Description
HOST NAME	Enter a desired CNC host name. On CNC Status Notification function, this host name is written as “Machine Name” in E-mail. If no host name is set, “NC-<MAC address>” is automatically set. Example of host name automatically set: NC-080019000001

MAIL SETTING screen

Press soft key [MAIL]. The MAIL SETTING screen is displayed.

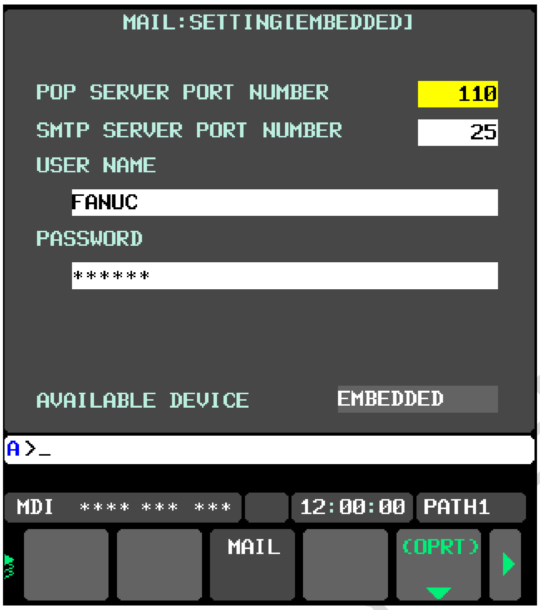


Fig.17.2.8.4 (c) MAIL SETTING screen

Setting items

Item	Description
POP SERVER PORT NUMBER	Specify the port number of POP server. Usually, set 110. If this value is 0, this value is treated as 110.
SMTP SERVER PORT NUMBER (NOTE 1)	Specify the port number of SMTP server. Usually, set 25. If this value is 0, this value is treated as 25.
USER NAME (NOTE 2)	Specify the user name to login to the POP server. A user name of up to 31 characters can be specified.
PASSWORD (NOTE 2)	Specify the password for the above user name. A password of up to 31 characters can be specified.

NOTE

1. SMTP is not actually used in this function.
   However, when creating a mail account, a portable terminal might check the SMTP setting. For this case, the SMTP server port number is necessary.
2. The user name and password of mail account in a portable terminal (POP client) have to match with these settings to login to CNC (POP server).
   However, if neither user name nor password is set in CNC, CNC does not check them when a portable terminal tries to connect to CNC. Thus in this case, the login to CNC always succeeds regardless of any user name and password in a portable terminal.

17.2.8.5 Related NC parameter

	#7	#6	#5	#4	#3	#2	#1	#0
14884	SNE	SNM	SNP	SNO				SNL

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When the following parameters are set, the power must be turned off before operation is continued.

#0 SNL When failing in login continuously five times from a portable terminal, CNC Status Notification function becomes:
0: Disabled.
1: Still enabled.

> **NOTE**
> When CNC Status Notification function has become disabled, please restart the CNC.

#4 SNO In CNC Status Notification function, the message mail is:
0: Not made.
1: Made.

#5 SNP The total number of machined parts, number of machined parts and number of parts required is:
0: Not included in the status mail.
1: Included in the status mail.

#6 SNM When the displayed language of CNC screen is Japanese, English or Simplified Chinese, the alarm message is:
0: Included in the alarm mail.
1: Not included in the alarm mail.

> **NOTE**
> When the displayed language of the CNC screen is not Japanese, English or Simplified Chinese, the alarm message is not included in the alarm mail regardless of this parameter. In this case, only the alarm number is reported.

#7 SNE CNC Status Notification function is:
0: Not used.
1: Used.

> **NOTE**
> If this parameter is not set, the setting screen of CNC Status Notification function is not displayed.

17.2.9 Setting of the Modbus/TCP Server Function

This subsection describes the setting required to operate the Modbus/TCP Server function with the embedded Ethernet function.

> ⚠ **WARNING**
>
> Only when a partner client device sends data and the Modbus/TCP Server function of CNC receives it, the data is checked and processed by CNC. Even if the communication of the client happens to stop unexpectedly, the Modbus/TCP Server function does not detect such situations.
>
> So, do not use the Modbus/TCP Server function for data exchange application concerned with machine safety.
>
> For example, even if the client attempts to send the "STOP" data at some timing, it is not guaranteed that it always reaches CNC. Thus, it is very dangerous to build the system concerned with machine safety by using Modbus/TCP Server function.
>
> If the data exchange concerned with machine safety fails, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

> **NOTE**
>
> 1 The Modbus/TCP Server function can also be used on the Fast Ethernet board. This subsection is written for the embedded Ethernet, but the basic content can be also applied when the Fast Ethernet board is used. For details on using the Fast Ethernet board, see "Industrial Ethernet CONNECTION MANUAL (B-64013EN)".
>
> 2 This function cannot be operated on the PCMCIA LAN card. It cannot be operated on a Fast Ethernet board and on embedded Ethernet at the same time. Switch it between them, using bit 1 (MOD) of parameter No.14882.
>
> 3 The communication speed of the embedded Ethernet is lower than a Fast Ethernet board.
>
> 4 The Modbus/TCP client function and the PROFINET IO controller function are not supported.

17.2.9.1 OVERVIEW

OVERVIEW

This section provides an overview of the Modbus/TCP Server function.

Modbus/TCP data model

The following four kinds of data formats, each called a "Table", are defined for the Modbus/TCP data model.

Table name	Object type (data type)	Type of READ/WRITE
Discrete input	Single bit	READ only
Coils	Single bit	READ-WRITE
Input Registers	16-bit word	READ only
Holding Registers	16-bit word	READ-WRITE

NOTE

1. For the CNC Modbus/TCP Server function, only "Holding Registers" can be used.
   "Discrete input", "Coils", and "Input Registers" cannot be used.
2. "READ" means the input of data as viewed from the Modbus/TCP client, and "WRITE" means the data of output as viewed from the client.

Modbus/TCP function codes

For the Modbus/TCP Server function of the CNC, the following function codes are supported.

Function code name	Code number
Read Holding Registers	03h
Write Multiple Registers	10h
Read/Write Multiple Registers	17h

Modbus area (Holding Registers)

For the Modbus/TCP Server function, the Modbus area (Holding Registers) of 65536 words (131072 bytes) is prepared.
And the Modbus area can be allocated to the PMC area.
Because of this, the data accessed from the Modbus/TCP client device to the Modbus area can be posted to the PMC area, and the user application (Ladder program etc.) can access the Modbus area through the PMC area.
This allocation can be set in up to three areas.
The Modbus/TCP area is a data area in word units, and can take address values of 1 to 65536.

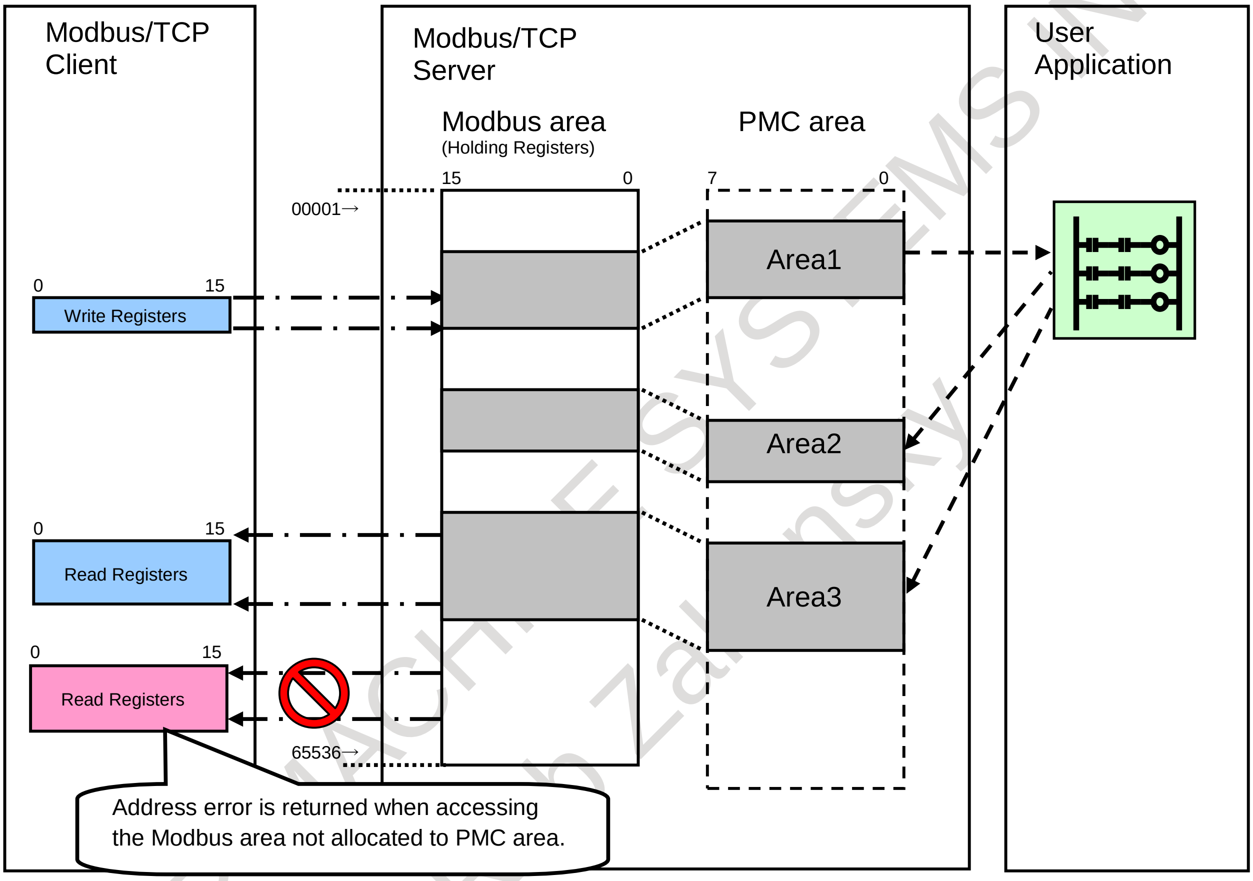


Fig.17.2.9.1 (a) Modbus area

NOTE

A Modbus/TCP client device can access only a Modbus area with a PMC area allocated. If a client device attempts to access a Modbus area with no PMC area allocated, an address error (Exception Code = 0x02, Exception Name = ILLIGAL DATA ADDRESS) is returned to the client device.
Be sure to allocate a PMC area to the Modbus area to be accessed by a client device.

SPECIFICATIONS of the Modbus/TCP Server Function

This section describes the specifications of the Modbus/TCP Server function of the CNC.

The following is a list of specifications of the Modbus/TCP Server function of the CNC.

Item	Description
Combination with other functions	The function can be operated at the same time as the function of embedded Ethernet.
PMC area that can be allocated	- DI : R, E, and D areas - DO : R, E, and D areas - Status : R and E area
Maximum DI/DO data size	Depends on the effective PMC area.
Port number (TCP)	502 (standard value)
Function specification	- "Automatic TCP connection management" is supported. This means that the user application (ladder program) need not take TCP connection into consideration. - "Non-priority connection pool" is supported. This means that if there are requests from more Modbus/TCP clients than a certain number, connections will be terminated in order of the oldest to the newest. (NOTE 1) ("Priority connection pool" is not supported.) - "Access control service" is not supported. This means that no client devices require authentication (user name or password) to access the FANUC Modbus/TCP Server function.
Maximum number of clients that can connect to a single CNC at the same time	3

NOTE

1 Use the function within the maximum number of clients that can be connected to a single CNC at the same time.
If there are connection requests from more clients than the maximum number, the communication with the oldest client will be disconnected. Thus, a client that wants to keep a connection must reconnect if disconnected, assuming that it may be disconnected from the server.

2 Settings of communication parameters are not subject to 8-level data protection function.

Allocating PMC Areas

To allocate a PMC area in the setting screen of each communication function, specify it as follows:

Input format) <Path number>:<PMC address>

For example, for R0500 on the second path of the PMC, input "2:R500".
If <Path number> is omitted (R500), the first path is assumed (1:R0500).
If the <:> key is not available, it can be substituted with the </> key or the <EOB> key. ":" is optional.
To clear "<Path number>:<PMC address>" previously set, input " " (blank). ("---" will be displayed). In this case, it is assumed that no PMC area is used.

> ⚠ **WARNING**
> Be careful not to cause an overlap between the PMC area to allocate and a PMC area used for other purposes. Immediately after setting all necessary communication parameters, check that the DI/DO data and status data are operated properly in a starter where safety is secured.
> If the machine is operated without confirmation of above explanation, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

> ⚠ **CAUTION**
> 1 Immediately after the power is turned on, all R area and E area as volatile memory in the PMC area are 0.
> 2 The D area in the PMC area is allocated to nonvolatile memory. In addition, the E area in the PMC area is normally allocated to volatile memory. However, it can also be used as nonvolatile memory.
> When the area is used as nonvolatile memory, the contents of the area are retained even after the power is turned off. So, special attention should be paid not to cause an unpredictable operation when the power is turned on next time.

Creating a Ladder Program

DI/DO data and status data allocated to a PMC area are processed by a ladder program.
The following provides notes on creating a ladder program required to construct a safety system.

> ⚠ **CAUTION**
> 1 The time after the power is turned on until communication is actually started may vary depending on the system configuration of the CNC and the state of the communication partner device.
> If it is necessary to strictly determine whether communication starts, it is not sufficient to determine it with time and status data only. Determine it with a method such as checking with actual DI data.
> 2 Create a ladder program in such a way that the system will be operated safely even if a communication error occurs.

Input signal (DI) and output signal (DO)

An output signal from the CNC is written by the ladder program into the PMC data area. The reading/writing process of a communication function reads the signal and sends it to the Ethernet network. An input signal also flows a similar route in the opposite direction.

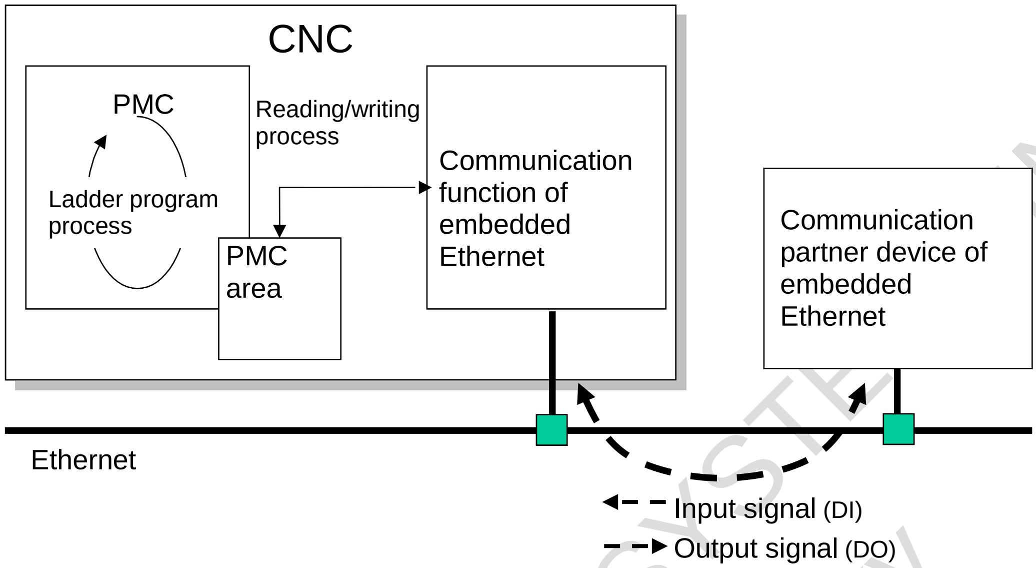


Fig.17.2.9.1 (b) Input signal (DI) and output signal (DO)

Ladder program process and reading/writing process by a communication function

Processing by the ladder program and the reading/writing process by a communication function operate asynchronously with one another.
Processing by the ladder program can operate independently of reading/writing process by a communication function, so the ladder program can be repeatedly executed at high-speed.

The following figure shows a time chart of the internal operation of the CNC with signals output from the ladder program.

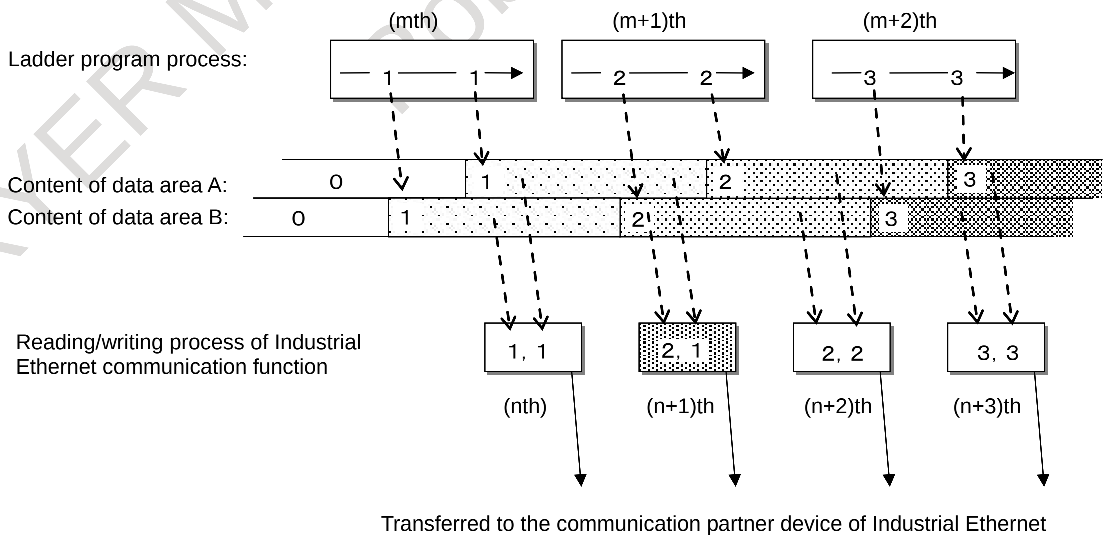


Fig.17.2.9.1 (c) Time chart

The upper part of the figure indicates that processing by the ladder program is performed periodically and writing to data area A or data area B is performed in the ladder program.

The middle part indicates how data area A and data area B in the PMC are updated in this case. There are differences in the timing in which data is written to data area A or data area B even in the same execution cycle of the ladder program, so data area A and data area B are not updated at the same time.

The lower part indicates that the reading/writing process by communication functions is cyclically made to read data in data area A and data area B. Since data area A and data area B are not updated at the same time, for example, in the case of (n+1)th refreshing, the data written by one execution of the ladder program cannot be read as one set.

Conversely, when input signals from communication functions are processed in the ladder program, the data written by one execution of writing may not be read by one execution of the ladder program.

> ⚠ **CAUTION**
>
> Reading/writing process by a communication function is made asynchronously with the execution of the ladder program. Therefore, keep the following in mind when creating a ladder program.
>
> <1> When an input signal from the communication function to be written in the specified PMC address is read from two points in the ladder program, even if the ladder program can be executed in one cycle, there is no guarantee that the same value can be read.
>
> <2> When the ladder program writes an output signal to the communication partner device in the specified PMC address, the signal may be transferred to the communication partner device before the ladder program is completely executed.

Unit of Data and Concurrency

The unit of I/O data can be word (2-bytes) only.
The concurrency of word (2-byte) data is guaranteed, and there will never be data segmentation. The concurrency of long (4-byte) data cannot be guaranteed.

> ⚠ **WARNING**
>
> If the data bigger than word (2-byte) data is used, data segmentation might occur.
> If word (2-byte) data is used, refer to “Creating a Ladder Program” for notes on creating a ladder program.
> If data segmentation occurs, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

To guarantee the concurrency of data, satisfy the following two conditions.

<1> In the ladder program, the following commands are used in units of two bytes.
MOVW, MOVN, XMOVB, SETNW, XCHGW, DSCHB, TBLRW, TBLWW, DSEQW, DSNEW, DSGTW, DSLTW, DSGEW, DSLEW, DMAXW, DMINW, EQW, NEW, GTW, LTW, GEW, LEW, RNGW, COMPB, EOR, AND, OR, NOT, EORW, ANDW, ORW, NOTW, SHLW, SHRW, ROLW, RORW, BSETW, BRSTW, BTSTW, BPOSW, BCNTW, CODB, DCNVB, DECB, TBCDW, FBCDW, ADDB, SUBB, MULB, DIVB, NUMEB, ADDSW, SUBSW, MULSW, DIVSW, MODSW, INCSW, DECSW, ABSSW, NEGSW

<2> When DI data or DO data is assigned to at the setting screen of a communication function, the value of items "ADDRESS" and "SIZE" are aligned with 2-byte boundaries.
Example) ADDRESS/SIZE 1:R0000/2, 2:R0002/4, 3:R0004/6, 1:E0002/8

Conversion of Endian

The byte arrangement in the Modbus area (Holding Registers) is little endian to make it easy for the ladder program to handle it.
With the appropriate setting item on the setting screen of the Modbus/TCP Server function, the byte arrangement can be changed to big endian.

> ⚠ **WARNING**
> Confirm that an appropriate value is set to the setting item of selecting the conversion of endian, and that the byte arrangement of the exchanged data is correct.
> If the machine is operated without confirmation of above explanation, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

Little endian

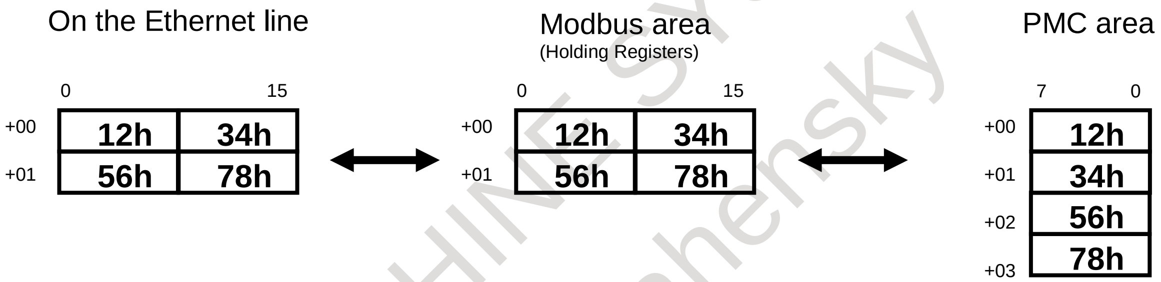


Fig.17.2.9.1 (d) Little endian

When Bit0 of OPTION1 is "1", byte arrangement of data becomes a big endian.

Big endian

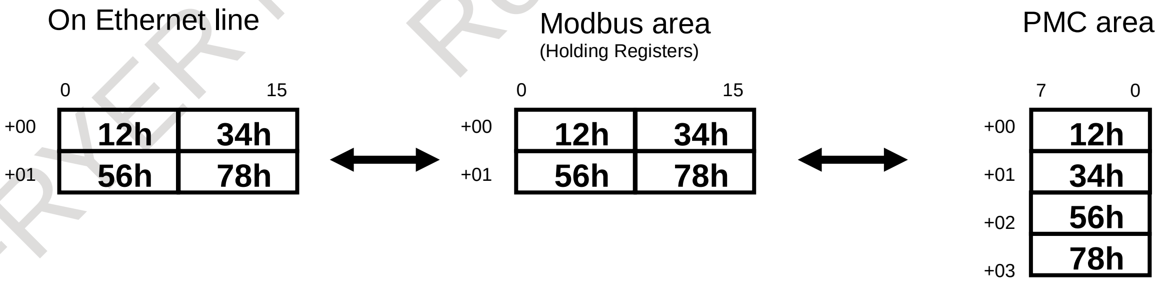
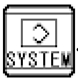


Fig.17.2.9.1 (e) Big endian

17.2.9.2 SETTING

This section describes the setting of the Modbus/TCP Server function.

Procedure

1 Press function key .

2 Soft key [EMBED PORT] appears. When the soft key does not appear, press the continue key.
3 Press soft key [EMBED PORT] to display the setting screen of the embedded Ethernet function.
4 Press soft key [COMMON] or [Modbus SET]. When the soft key does not appear, press the continue key.

COMMON Screen

In the above-mentioned procedure, press soft key [COMMON] to display the COMMON screen (BASIC).

Fig.17.2.9.2 (a) COMMON screen (BASIC)

Setting items

Item	Description
IP ADDRESS	Specify the IP address of the embedded Ethernet. (Example of specification format: "192.168.0.100")
SUBNET MASK	Specify the mask address for the IP addresses of the network. (Example of specification format: "255.255.255.0")
ROUTER IP ADDRESS	Specify the IP address of the router. Specify this item if the network contains a router. (Example of specification format: "192.168.0.253")

Display item

Item	Description
MAC ADDRESS	MAC address of the embedded Ethernet.

Setting Screen of the Modbus/TCP Server Function

In the above-mentioned procedure, press soft key [Modbus SET] to display the setting screen of the Modbus/TCP Server function.

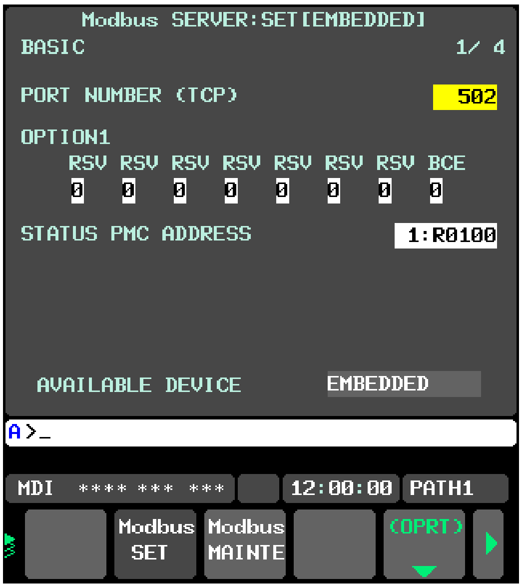


Fig.17.2.9.2 (b) Setting screen 1 (BASIC)

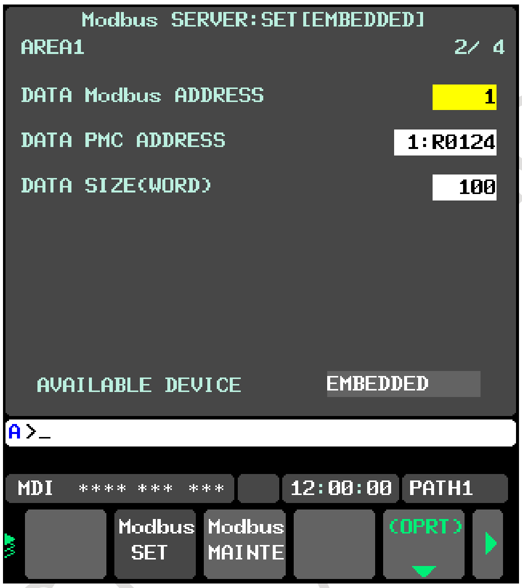


Fig.17.2.9.2 (c) Setting screen 2 (areas 1 to 3)

Setting items

Item	Description
PORT NUMBER (TCP)	Specify the port number for using the Modbus/TCP Server function. The setting range is from 0 to 65535. Usually, set 502. When 0 is set, the Modbus/TCP Server function is not operated.
OPTION1	Bit 0: BCE The byte arrangement in the Modbus area is assumed to be 0: little endian 1: big endian Bits 1 to 7: RSV Be sure to set 0.
STATUS PMC ADDRESS	Specify the top address in the E/R area of the PMC in which to store the status. The setting range depends on the effective PMC area. A 1-byte area is used.
DATA Modbus ADDRESS	Specify the top address in the Modbus area (Holding registers) used to exchange I/O with the Modbus/TCP client. The setting range is 1 to 65536. For details of the endian of the data in the Modbus area (Holding Registers), see Subsection 18.2.6.1, "Conversion of Endian".
DATA PMC ADDRESS	Specify the top address in E/R/D area of the PMC used to exchange I/O with the Modbus/TCP client. The setting range depends on the effective PMC area. Only the even-numbered address can be set.
DATA SIZE	Specify the data size (unit: word size) used to exchange I/O with the Modbus/TCP client. The setting range depends on the effective PMC area. If not exchange I/O with the area, specify 0 for this parameter.

> ⚠ **WARNING**
> When setting PMC address for Modbus/TCP server function, thoroughly confirm the content of "GENERAL WARNINGS FOR CNC APPLICATION DEVELOPMENT" in "SAFETY PRECAUTIONS" at the beginning of this manual.
> Be careful not to cause an overlap between the PMC area to allocate and a PMC area used for other purposes.
> If the machine is operated without confirmation of above explanation, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

> **NOTE**
> 1 For the changed parameters to take effect, turn the CNC power off and back on or restart embedded Ethernet at the COMMON screen.
> 2 Communication parameters of the Modbus/TCP Server function that is operated on embedded Ethernet are included in the embedded Ethernet parameters.
> 3 Set the Modbus areas for areas 1 to 3 so that they do not overlap. If any overlap, different actions are taken for READ and WRITE, as follows:
> For READ, area n where n is the largest is given priority, and data is read from that area.
> For WRITE, the same value is written to all the overlapping areas.

Status

The status (1 byte) is detected by the Modbus/TCP Server function.

		Status							
		#7	#6	#5	#4	#3	#2	#1	#0
Status PMC address	**+0**	**COM**	**INT**	**-**	**-**	**-**	**-**	**-**	**ERR**

Bit	Name	Description
#0	ERR	Unrecoverable error 0: The Modbus/TCP Server function is operating correctly. 1: An unrecoverable error occurred in the Modbus/TCP Server function.
#1-5	-	Reserved
#6	INT	Initialization completion 0: The initialization of the Modbus/TCP Server function is not completed. 1: The initialization of the Modbus/TCP Server function is completed.
#7	COM	Communicating 0: Not communicating with a Modbus/TCP client. 1: Communicating with one or more Modbus/TCP clients.

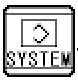
17.2.9.3 MAINTENANCE

This section describes the maintenance of the Modbus/TCP Server function.

Maintenance Screen of the Modbus/TCP Server Function

The connection status of the client connected to the Modbus/TCP Server can be checked on this screen.

Procedure

1. Press function key 

.
2. Soft key [EMBED PORT] appears. When the soft key does not appear, press the continue key.
3. Press soft key [EMBED PORT], and then soft key [Modbus MAINTE] appears on the setting screen of the Ethernet function.
4. Press soft key [Modbus MAINTE], and then the maintenance screen (CLIENT LIST) of the Modbus/TCP Server function is displayed.

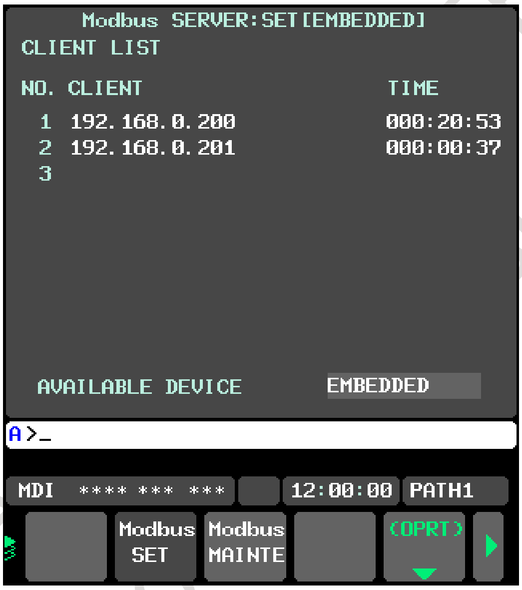


Fig.17.2.9.3 (d) Maintenance screen (CLIENT LIST) of the Modbus/TCP Server function

Display items

Item	Description
CLIENT	The IP addresses of the Modbus/TCP clients connected to this Modbus/TCP Server are displayed. (Example of display format: "192.168.0.200") They are displayed in the order in which the clients are connected.
TIME	The connecting times of the Modbus/TCP clients connected to this Modbus/TCP Server are displayed. The displayed times are updated automatically. The display format for a time of hour (hhh), minute (mm) and second (ss) is "hhh:mm:ss". The maximum value is "999:59:59", and if the maximum value is exceeded, the time will not be updated from "999:59:59".

17.2.9.4 RELATED NC PARAMETER

	#7	#6	#5	#4	#3	#2	#1	#0
14882							MOD	

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#1 MOD The Modbus/TCP Server function is used
- 0: on a Fast Ethernet.
- 1: on embedded Ethernet (embedded Ethernet port).

17.3 SWITCHING BETWEEN THE EMBEDDED ETHERNET DEVICES

There are two types of embedded Ethernet devices: the embedded Ethernet port and PCMCIA Ethernet card.
Screen operation is required to switch between these two types of devices.

Procedure

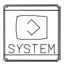
1. Press the function key SYSTEM.
2. Soft keys [EMBED] and [PCMCIA LAN] appear.
   (When there is no soft keys, press the continue key.)
3. Press soft key [EMBED] or [PCMCIA LAN], press soft key [COMMON], and then press [(OPRT)] to display soft key [EMB/PCMCIA].
4. Pressing soft key [EMB/PCMCIA] switches between enabled devices.

> **NOTE**
> 1 Information on a switched device is stored in nonvolatile memory.
>   On the next power-on, the device last selected can be used as is.
> 2 When using the unsolicited messaging function, note the following:
>   - Processing is forcibly started from the "Not Ready" state, regardless of the unsolicited message transfer state.
>   - When the parameter for control is set to use the response notification method, RES and RES_CODE (0x01) are posted to the ladder program.
>   - When the parameter for control is set to use the simplified method, REQ is cleared.

17.4 RESTART OF THE EMBEDDED ETHERNET

Communication using the embedded Ethernet can be restarted.

Procedure

1. Press the function key  .
2. Soft keys [EMBED] and [PCMCIA] appear.
   (When there is no soft keys, press the continue key.)
3. Press soft key [EMBED] or [PCMCIA LAN], press soft key [COMMON], and then press [(OPRT)] to display soft key [EMB/PCMCIA].
4. Pressing soft key [RSTART] resets embedded Ethernet communication and then restarts it.

> **NOTE**
> 1. Pressing soft key [RSTART] forcibly interrupts communication even when it is in progress.
> 2. This function makes a restart by software. An actual restart may be impossible under some conditions.
> 3. When using the unsolicited messaging function, note the following:
>   - Processing is forcibly started from the "Not Ready" state, regardless of the unsolicited message transfer state.
>   - When the parameter for control is set to use the response notification method, RES and RES_CODE (0x01) are posted to the ladder program.
>   - When the parameter for control is set to use the simplified method, REQ is cleared.

17.5 MAINTENANCE SCREEN FOR EMBEDDED ETHERNET FUNCTION

With the embedded Ethernet function, a dedicated maintenance screen is available.
The maintenance screen enables operations to be checked when the embedded Ethernet function operates abnormally.

Displaying and operating the PING screen

Procedure

1. Press the function key  .
2. Soft keys [EMBED] and [PCMCIA LAN] appear.
   (When there is no soft keys, press the continue key.)
3. By pressing the [EMBED] soft key, the Ethernet Setting screen for the embedded Ethernet is displayed.
   By pressing the [PCMCIA LAN] soft key, the Ethernet Setting screen for the PCMCIA Ethernet card can be set.
4. Press soft key [PING] and then press [(OPRT)].
5. To send the PING command to connection destination 1 for FTP file transfer, press soft key [PING FTP1]. Similarly, to send the PING command to connection destination 2 or 3, press [PING FTP2] or [PING FTP3], respectively.

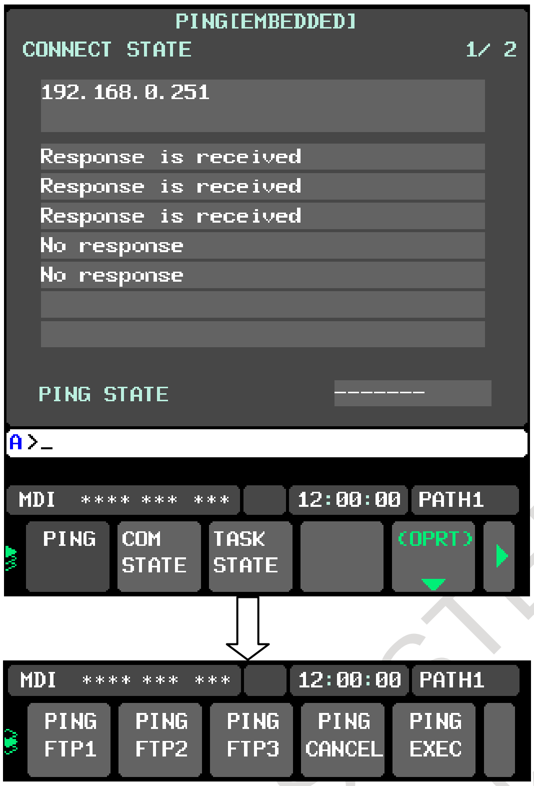
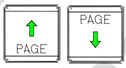


Fig. 17.5 (a) PING connection status screen

6 To send the PING command to the desired destination, enter the address of the destination on the PING setting screen (Fig. 17.5(b)). (Page keys  are used for switching.)

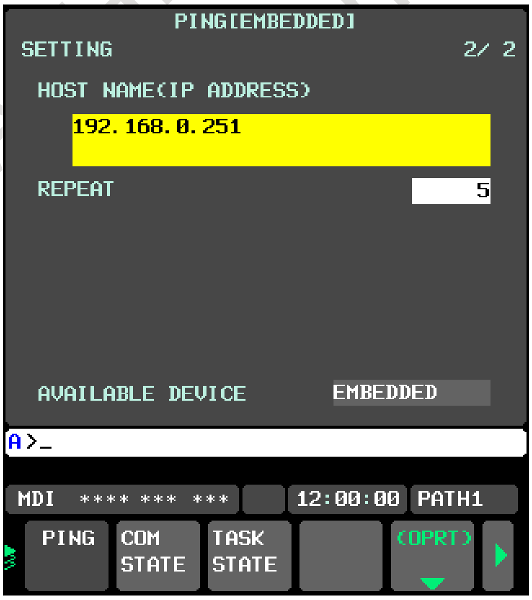


Fig. 17.5 (b) PING connection status screen

7 After entering the address and the repeat count, press the soft key [PING]. The specified number of PING commands are sent to the specified destination.
8 To cancel the PING command currently being sent, press soft key [PING CANCEL].

Displaying Communication status screen

Procedure

1. Press the function key SYSTEM.
2. Soft keys [EMBED] and [PCMCIA LAN] appear.
   (When there is no soft keys, press the continue key.)
3. By pressing the [EMBED] soft key, the Ethernet Setting screen for the embedded Ethernet is displayed.
   By pressing the [PCMCIA LAN] soft key, the Ethernet Setting screen for the PCMCIA Ethernet card can be set.
4. To display the communication status (Fig. 17.5(c)) of the embedded Ethernet, press soft key [COM STATE].
   Page keys PAGE PAGE can be used to switch between the sending state and the receiving state.

Fig. 17.5 (c) Communication status screen

Display items

Item	Description
BAUDRATE	Displays the baud rate and transmission method. Transmission rate: 100Mbps or 10Mbps Transmission method: Full duplex or half duplex -------- : Not connected to HUB
SEND PACKET	Displays the number of transmitted packets.
COLLISION CARRIER SENSE LOST DELAYOVER UNDERRUN SEND PARITY ERROR	Displays the number of errors detected during transmission of packets.
RECEIVE PACKET	Displays the number of packets received.
ALIGNMENT ERROR CRC ERROR OVERRUN ERROR FRAME LENGTH ERROR RECV PARITY ERROR	Displays the number of errors detected during reception of packets.
AVAILABLE DEVICE	Currently available device of embedded Ethernet Displays either the embedded Ethernet port or the PCMCIA Ethernet card.

TASK STATE screen

Procedure

1 Press the function key SYSTEM.

2 Soft keys [EMBED] and [PCMCIA LAN] appear.
(When there is no soft keys, press the continue key.)

3 To display the Ethernet Setting screen for the embedded Ethernet port or the PCMCIA Ethernet card, press soft key [EMBED] or [PCMCIA LAN], respectively.

4 Pressing soft key [TASK STATUS] causes the task status (Fig. 17.5(d)) of the embedded Ethernet function to be displayed.

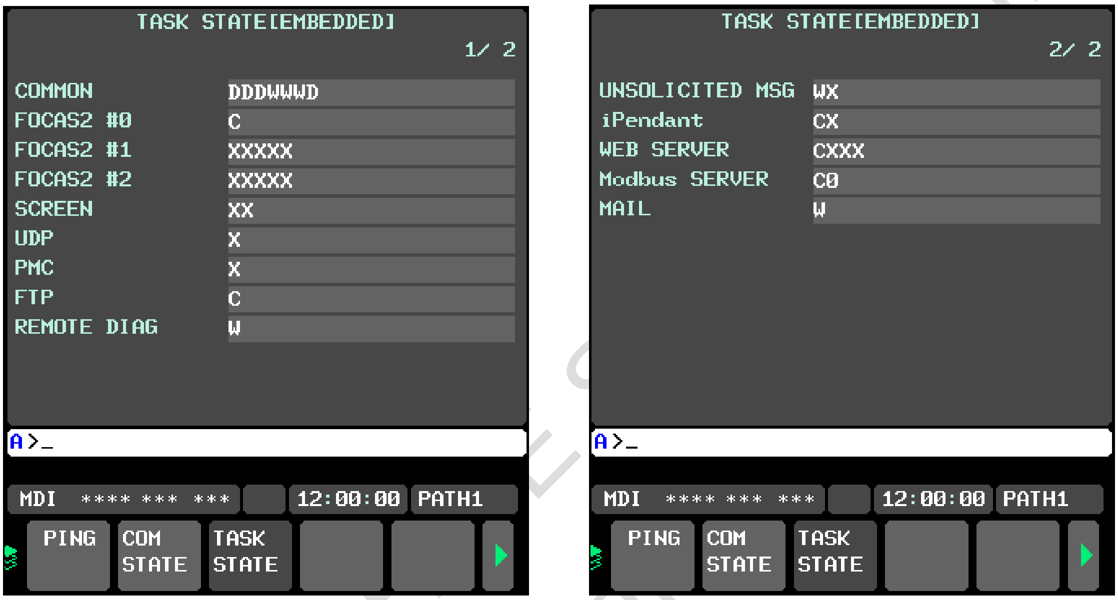


Fig. 17.5 (d) TAST STATE screen

Table 17.5 (a) lists the meaning of each symbol.

Table 17.5 (a)

	Symbol and meaning
FOCAS2 #0	C: Waiting for a connection from the host W: Data processing in progress (1) D: Data processing in progress (2) N: FOCAS2 out of service
FOCAS2 #1,#2	W: Data processing in progress (1) D: Data processing in progress (2) X: Not yet executed
SCREEN	W: Data processing in progress (1) D: Data processing in progress (2) X: Not yet executed
UDP	W: Data processing in progress (1) D: Data processing in progress (2) X: Not yet executed
PMC	W: Data processing in progress (1) D: Data processing in progress (2) X: Not yet executed
FTP	C: Execution wait W: Data processing in progress (1) D: Data processing in progress (2) X: Not yet executed

	Symbol and meaning
REMOTE DIAG	W: Data processing in progress (1) D: Data processing in progress (2) X: Not yet executed
UNSOLICITED MSG	W: Data processing in progress (1) D: Data processing in progress (2) N: Abnormal state X: Not yet executed Number: Alive signal (UDP) transmission in progress when count-up operation is performed
iPendant	C: Waiting for a connection from iPendant W: Data processing in progress (1) D: Data processing in progress (2) X: Not yet executed
WEB SERVER	C: Waiting for a connection W: Data processing in progress (1) D: Data processing in progress (2) X: Not yet executed
Modbus SERVER	C: Waiting for a connection W: Data processing in progress (1) D: Data processing in progress (2) X: Not yet executed Number: Number of sockets currently connected
MAIL	W: Data processing in progress (1) D: Data processing in progress (2) X: Not yet executed

17.6 LOG SCREEN OF THE EMBEDDED ETHERNET FUNCTION

This screen displays the log of the embedded Ethernet function.

NOTE
1 If alarm SR2032, “EMBEDDED ETHERNET/DATA SERVER ERROR” is issued during data transfer using the embedded Ethernet function, check the error details on the log screen of the embedded Ethernet function.
2 The log information for the embedded Ethernet function is stored in volatile memory and it is lost when the power of CNC is turned off. If an error occurs, check the log information before turning the power off.

Displaying the log screen

Procedure

1 Press the function key .

2 To display the log screen (Fig. 17.6(a)) for the embedded Ethernet port or PCMCIA Ethernet card, press soft key [EMBED LOG] or [PCMCIA LOG], respectively. (When there is no soft keys, press the continue key.)

Fig. 17.6 (a) LOG screen

The newest error log appears at the top of the screen. The date and time when an error occurred are displayed at the right end of the line. The format of date and time data is “MMM.DD hh:mm:ss” where MMM represents a month, dd represents a day, hh represents hours, mm represents minutes, and ss represents seconds.
The date and time of the upper item shown Fig. 17.6(a) is October 16, 12:57:06.
To clear the log, press soft keys [(OPRT)] and [CLEAR] in that order. (Fig. 17.6 (b))

Fig. 17.6 (b)

The log for each function can be displayed by using soft keys on the embedded Ethernet log screen.

(1) Soft key [ALL]
Displays all log related to the embedded Ethernet.

(2) Soft key [COMMON]
Displays the log related to the parameter settings of the embedded Ethernet function and the basic communication function.

(3) Soft key [FOCAS2]
Displays the log related to the FOCAS2/Ethernet function.

(4) Soft key [FTP TRANS]
Displays the log related to FTP file transfer.

(5) Soft key [UNSOLT MSG]
Displays the log related to the unsolicited messaging function.

(6) Soft key [WEB]
Displays the log related to the CNC screen Web server function.

(7) Soft key [Modbus]
Displays the log related to the Modbus/TCP Server function.

(8) Soft key [REMOTE DIAG]
Displays the log related to the Machine Remote Diagnosis.

(9) Soft key [MAIL]
Displays the log related to the CNC Status Notification function.

Error and message

Error No.	Log message	Description and necessary action
E-0118 E-0119	Error occurred while wait for FOCAS2 pdu	A communication error has occurred because of any of the following: → The network quality has been lowered to such a level that data cannot be received from a PC at the other end. The communication channel has been logically shut down. → Software running on a PC at the other end has logically shut down the communication channel. → The Ethernet cable has been disconnected.
E-011A	All communication paths are busy	All the FOCAS2/Ethernet communication channels are busy.
E-0126	No response from RMT DIAG server	The IP address of the machine remote diagnosis accepting server may be invalid or the power to the machine remote diagnosis accepting server may be off. Check whether the IP address of the machine remote diagnosis accepting server is valid and whether the power to the machine remote diagnosis accepting server is on. Alternatively, the machine remote diagnosis accepting server may not respond to the PING command to increase the security level (such as a firewall setting). Set bit 1 of parameter No.14880 to 1 and connect the server again.
E-0148	Cannot save parameter for Unsolicited Message	When the FOCAS2 function cnc_wrunsolicprm2 was received, the parameter for the unsolicited messaging function could not be saved for one of the following causes: → The mode of the unsolicited messaging function is not set to "PC mode". → The state of the unsolicited messaging function is not "Not Ready". → The argument "parameter-for-unsolicited-message" of the FOCAS2 function cnc_wrunsolicprm2 includes an invalid value.

Error No.	Log message	Description and necessary action
E-0149	The received parameter for Unsolicited Message is wrong	When the FOCAS2 function cnc_wrunsolicprm2, cnc_rdunsolicprm2, cnc_unsolicstart, or cnc_unsolicstop was received, the argument "parameter-number-for-unsolicited-message" was found to be invalid.
E-0200	(Received message from FTP server)	A message sent by the FTP server is displayed as it is. A message containing characters other than ASCII characters may not be displayed correctly.
E-0202	Connection failed with FTP server	Software of the FTP server may not be running. Run the FTP server software. Alternatively, the FTP server may not respond to the PING command to increase the security level (such as a firewall setting). Set bit 1(PCH) of parameter No. 14880 to "1" and connect the server again.
E-0207	The router is not found	The specified IP address of the router may be wrong. Alternatively, the router may be turned off. Check whether the IP address of the router has been correctly specified and whether the router is turned on.
E-0208	The FTP server is not found	The specified IP address of the FTP server may be wrong. Alternatively, the FTP server may be turned off. Check whether the IP address of the FTP server has been correctly specified and whether the FTP server is turned on. Alternatively, the FTP server may not respond to the PING command to increase the security level (such as a firewall setting). Set bit 1(PCH) of parameter No. 14880 to "1" and connect the server again.
E-020B	Cannot login into FTP server	Check whether a correct user name and password are specified when logging into the FTP server.
E-020C	The parameters of FTP server are wrong	Check whether a correct user name and password are specified when logging into the FTP server.
E-020D	Changing a work folder of host failed	Check the work folder logging into the FTP server.
E-041A	Frame transmission failed (TCP)	A communication error has occurred because of any of the following: → The network quality has been lowered to such a level that data cannot be received from a PC at the other end. The communication channel has been logically shut down. → Software running on a PC at the other end has logically shut down the communication channel. → The Ethernet cable has been disconnected. → Data cannot be posted to the communication destination due to a firewall setting.
E-0901	Cannot read MAC address	The MAC address is not written in the hardware. Alternatively, the hardware has been damaged.
E-0A06	Network is too busy	An excessive amount of data is flowing over the network. One possible solution is to divide the network.
E-0B00	The own IP address is wrong	Specify a correct IP address in the designated format.
E-0B01	The own IP address is not set	Specify an IP address.
E-0B02	Subnet mask is wrong	Specify a correct subnet mask in the designated format.
E-0B03	Subnet mask is not set	Specify a subnet mask.
E-0B04	Router IP address is wrong	There may be class disagreement between the IP address of the local node and the IP address of the router.
E-0B05	IP address of DNS server is wrong	There may be class disagreement between the IP address of the local node and the IP address of the DNS server.
E-0B06	The own host name is wrong	Check whether a correct host name is specified.
E-0B07	The own domain name is wrong	Check whether a correct domain name is specified.

Error No.	Log message	Description and necessary action
E-0B08	TCP port number is wrong	A value beyond the permissible setting range may be specified.
E-0B09	UDP port number is wrong	A value beyond the permissible setting range may be specified.
E-0B0B	IP address of remote FTP server is wrong	Specify a correct IP address in the designated format.
E-0B0C	Port NO of a remote FTP server is wrong	A value beyond the permissible setting range may be specified.
E-0B0D	User name of remote FTP server is wrong	The specified user name may contain a prohibited character.
E-0B0E	Password of remote FTP server is wrong	The specified password may contain a prohibited character.
E-0B0F	Login folder of remote FTP srv is wrong	The specified log-in folder name may contain a prohibited character.
E-0B18	Cannot set because DHCP is available	To allow a set-up, disable the DHCP client function.
E-0B19 E-0B1A	Embedded Ethernet hardware isn't found	The software or hardware of embedded Ethernet function cannot be recognized. Check whether the software has been incorporated. Check whether the hardware is sound.
E-0B27	Unsolicited Message isn't available	The software condition for using the unsolicited messaging function is not satisfied. The cause may be one of the following: → parameters for using the unsolicited messaging function are not set. For the parameters, see Subsection 18.2.4, "Setting of the Unsolicited Messaging Function".
E-0B29	Mode of Unsolicited Message is wrong	In the CNC mode, the FOCAS2 function cnc_wrunsolicprm2 cannot be executed.
E-0B2A	Status of Unsolicited Message is wrong	The state of the unsolicited messaging function was other than "Not Ready", so that the parameters for the unsolicited messaging function could not be updated. The cause may be one of the following: → In a state other than "Not Ready", the FOCAS2 function cnc_wrunsolicprm2 or cnc_unsolicstart was executed. → In a state other than "Not Ready", soft key [APPLY] was pressed.
E-0B2B	Cannot refresh parameter of Unsolicited Message	The parameters for the unsolicited messaging function could not be updated. The cause may be one of the following: → The problem of E-0B29 or E-0B2A occurred. → A parameter for the unsolicited messaging function includes an invalid value.
E-0B44	Invalid value exists in Transmission parameter of Unsolicited Message	The parameter for the unsolicited messaging function, TRANSMISSION NUMBER or TRANSMISSION PARAMETER (NO. 1 to NO. 3), includes an invalid value.
E-0B45	The total of Transmission size of Unsolicited Message exceeds the limitation	The sum of sizes specified by the parameters for the unsolicited messaging function, TRANSMISSION PARAMETER NO. 1 to NO. 3, exceeds the maximum specifiable number of bytes. For the maximum specifiable number of bytes, see the setting item "TRANSMISSION PARAMETER" in Subsection 18.2.4.4, "Setting on the CNC screen".
E-0B47	TCP port number of Modbus/TCP Server is wrong	Zero is specified. To use this function, specify TCP port number.
E-0B48	Status PMC address of Modbus/TCP Server is out of range	A value beyond the permissible setting range is specified. Check the status PMC address settings.
E-0B49	Data PMC address of Modbus/TCP Server is wrong	An odd-numbered address is specified. Check the data PMC address setting.

Error No.	Log message	Description and necessary action
E-0B4A	Data PMC address of Modbus/TCP Server is out of range	A value beyond the permissible setting range is specified. Check the data PMC address setting or data size setting.
E-0B4B	Modbus area of Modbus/TCP Server is out of range	A value beyond the permissible setting range is specified. Check the data Modbus address setting or data size setting.
E-1001	All Modbus communication paths are busy	The number of connected communication parties (Modbus/TCP clients) exceeded the maximum number of connectable clients. Connections to communication parties (Modbus/TCP clients) will be terminated in order of the oldest to the newest.
E-1003	Version number of Modbus packet is wrong	Specify a correct version number according to the protocol of Modbus/TCP.
E-1004	Length of Modbus packet is wrong	Specify a correct size according to the protocol of Modbus/TCP.
E-100B	Function code of Modbus packet is wrong	The specified function code may not be supported. For details of supported function codes, see "Modbus/TCP function codes" in Subsection 18.2.6.1, "OVERVIEW".
E-1015 E-1016	Data address of Modbus packet is wrong	Specify a correct data address according to the protocol of Modbus/TCP.
E-1017 E-1018 E-1019	Data value of Modbus packet is wrong	Specify a correct data address according to the protocol of Modbus/TCP.
E-101A	Data address that client requested is out of range (1-3)	No PMC area is assigned to the Modbus area requested from the connected communication party (Modbus/TCP client). Assign a PMC area.
E-1300	(Sent message to Web browser)	It is a message transmitted to notify the Web browser of the error detected with the Web server.
E-1301	User name is not set	Specify a user name.
E-1302	Password is not set	Specify a password.
E-1303	User name is wrong	Check whether a correct user name.
E-1304	Password is wrong	Check whether a correct password.
E-1305	All communication paths are busy	All the Web communication channels are busy.
E-1306	Session ended	The connection with the Web browser ended.
E-1307	Session canceled	The connection with the Web browser aborted compulsorily by either of the following factor. → The Web browser connection prohibition signal WBEND <G0579.5> became “1”. → CNC screen display function started on other personal computer.
E-1308	Session started	The connection with the Web browser started.
E-1309	Login failed	The CNC screen Web server function became disabled. Turn off / on the power of the CNC.
E-1400	(Sent message to POP client)	It is not generated usually.
E-1401	(Sent message to SMTP client)	It is not generated usually.
E-1402	User name is not set	Enter user name of CNC Status Notification.
E-1403	Password is not set	Enter password of CNC Status Notification
E-1404	User name is wrong	Confirm user name of CNC Status Notification.
E-1405	Password is wrong	Confirm password of CNC Status Notification
E-1406	Login failed	CNC Status Notification function has not been usable. Please restart the CNC.
E-XXXX	(No message)	An internal error has occurred. Make a notification of the error number.

18 DIAGNOSIS FUNCTION

18.1 SERVO WARNING INTERFACE

Overview

The servo system can report the warning status before one of the following target alarms occurs.
When the warning status is entered, a report to the PMC is issued. For example, this signal can be used by the machine for retracting tools from the time a warning occurs by the time a servo alarm occurs.

Signal

Servo warning detail signals SVWRN1 to 4 <Fn093.4 to Fn093.7>

[Classification] Output signal

[Function] Reports the warning signal corresponding to the state of the servo amplifier.

[Output condition] The Table 18.1(a) shows the warning statuses of the servo amplifier and their corresponding warning signals.

Table 18.1 (a)

Corresponding alarm messages	Warning status signals (F93)				Time from when a warning state signal is issued to until an alarm occurs
	SVWRN4 (#7)	SVWRN3 (#6)	SVWRN2 (#5)	SVWRN1 (#4)	
SV0444: SV INTERNAL FAN FAILURE	1	0	0	0	One minute
SV0601: SV EXTERNAL FAN FAILURE	1	0	0	1	Until overheat occurs (inconstant)
SV0040: SV EXTERNAL FAN FAILURE	1	0	1	1	Depending on where the trouble occurs in
SV0443 : SV INTERNAL FAN FAILURE	1	1	0	0	One minute
SV0606: PS EXTERNAL FAN FAILURE	1	1	0	1	Until overheat occurs (inconstant)
SV0431: PS OVERLOAD	1	1	1	0	One minute
SV0607: PS IMPROPER INPUT POWER	1	1	1	1	PSMR: Five seconds, PSM: One minute

A timing chart for handling a warning is shown Fig. 18.1.

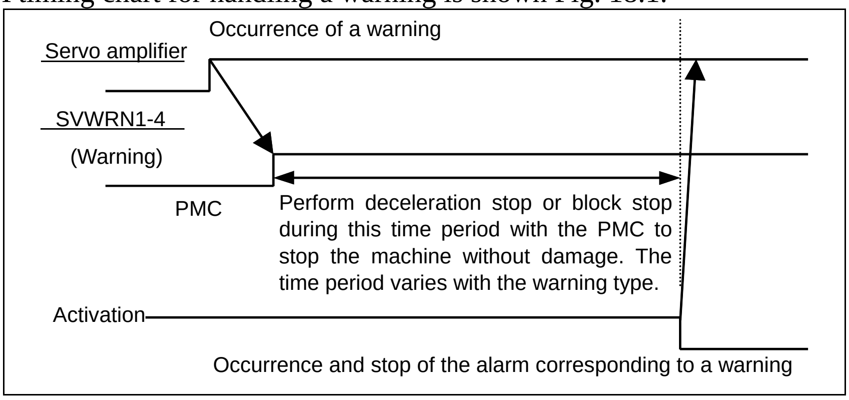


Fig. 18.1 (a)

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn093	SVWRN4	SVWRN3	SVWRN2	SVWRN1				

Warning status alarms for servo amplifiers

When bit 2 (SWP) of parameter No. 1807 is set to 0, if a servo amplifier is placed in the warning status, a DS alarm is output in addition to the servo warning detail signals.

If one of the following alarms is issued, automatic operation enters the feed hold state and movements along all controlled axes including PMC axes are decelerated to a stop. Issuing a warning status alarm does not cause to the servo motor to be deactivated.

Table 18.1 (b)

Alarm No. when the servo amplifier is in the warning status	Alarm No. when the servo amplifier is in the alarm status	Alarm message	Operation performed when the warning status occurs
DS0608	SV0444	n-axis : SV INTERNAL FAN FAILURE	Immediately after the fan stops, alarm DS0608 is displayed and movements along axes enter the feed hold stop state. The servo motor is kept active for 1 minute. After that, the servo motor is deactivated and alarm SV0444 is displayed.
DS0609	SV0601	n-axis : SV EXTERNAL FAN FAILURE	Immediately after the fan stops, alarm DS0609 is displayed and movements along axes enter the feed hold stop state. When bit 2 (SWP) of parameter No. 1807 is set to 1 after that, the alarm display can be canceled by a reset and operation can be restarted.
DS0610	SV0443	n-axis : SV INTERNAL FAN FAILURE	Immediately after the fan stops, DS0610 is displayed and movements along axes enter the feed hold stop state. The servo motor is kept active for 1 minute. After that, the servo motor is deactivated and alarm SV0443 is displayed.
DS0611	SV0606	n-axis : PS EXTERNAL FAN FAILURE	Immediately after the fan stops, DS0611 is displayed and movements along axes enter the feed hold stop state. When bit 2 (SWP) of parameter No. 1807 is set to 1 after that, the alarm display can be canceled by a reset and operation can be restarted.
DS0612	SV0431	n-axis : PS OVERLOAD	
DS0613	SV0607	n-axis : PS IMPROPER INPUT POWER	
DS0614	SV0040	n-axis : PS EXTERNAL INPUT COMPONENT ERROR	Alarm DS0614 is displayed and movements along axes enter the feed hold stop state. The servo motor is kept active for fixed time. The time until deactivation depends on where the trouble occurs in.

After an alarm in the warning status listed in Table 18.1 (b) is issued, when the αi servo amplifier is actually placed in the alarm status, the servo motor is deactivated and the number of an alarm in the alarm status is additionally displayed.

When bit 2 (SWP) of parameter No. 1807 is set to 1, if a servo amplifier is placed in the warning status, only the servo warning detail signals are output and no alarm in the warning status (alarm DS0608 to DS0614) is issued. If a servo amplifier is placed in the warning status during automatic operation, automatic operation is kept. So, decelerate the movements along axes to a stop using the servo warning detail signals before the servo motor is deactivated. Note that if the movements along axes are not decelerated to a stop, the servo motor is suddenly deactivated during movement along axes.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1807						SWP		

[Input type] Parameter input
[Data type] Bit path

#2 SWP This parameter specifies the operation of the αi series servo amplifier in its warning state (for example, with the fan stopped).

0: An alarm is issued when the amplifier is placed in the warning state. Automatic operation enters the feed hold state and the servo axis is decelerated to a stop.
1: An alarm is not issued even when the amplifier is placed in the warning state. Automatic operation is kept. The servo is deactivated if the amplifier shifts from the warning state to the alarm state.

> ⚠ **CAUTION**
> If operation is continued with the external fan stopped while bit 2 (SWP) of parameter No. 1807 is set to 1, the servo amplifier may be overheated and "overheat alarm", "IPM alarm", or "VRDY off alarm" may be issued depending on the operating conditions. If such an alarm is issued, the amplifier is deactivated and the servo motor is stopped by the dynamic brake, involving a risk of breaking the workpiece or tool as the stop from high-speed rotation requires an extended distance. The user should therefore understand that the operation with bit 2 (SWP) of parameter No. 1807 set to 1 is a temporary step to take before fan replacement. Once the fan has stopped, be sure to replace the fan immediately and set bit 2 (SWP) of parameter No. 1807 back to 0.
>
> If bit 2 (SWP) of parameter No. 1807 is set to 1, the warning text "FAN" blinks on the NC screen to show that the external fan has been stopped. Also on the machine side, monitor the warning signal output to the PMC and remind the operator of the operation with the fan stopped.

18.2 SPINDLE WARNING INTERFACE

Overview

The warning state can be reported before an alarm is issued. When the warning state is entered, a report to the PMC is sent.

For example, this signal can be used for retracting tools or reducing cutting load from the time a warning occurs by the time an overheat alarm occurs. In addition, diagnosis data also contains warning numbers.

Signal

Spindle warning detailed signals SPWRN1 to SPWRN9 <Fn264.0 to 7, Fn265.0>

[Classification] Output

[Function] Reports the warning number corresponding to the state of the αi spindle amplifier.

[Output condition] When the αi spindle is in the warning state, a warning number consisting of SPWRN1 to SPWRN9 is output as nine-bit binary data.

If warnings occurred on multiple αi spindle amplifiers, the warning number of the ai spindle having the smallest axis number is output.

The warning numbers and their descriptions are shown Table 18.2.

Table 18.2

Warning number	Contents	Details
56	Internal fan stopped	If the internal fan stops, the warning signal is output. Since the spindle continues to operate at this time, use the PMC to perform processing as needed. About one minute after the warning signal is output, an alarm occurs.
88	Radiator cooling fan stopped	If the radiator cooling fan stops, the warning signal is output. Since the spindle continues to operate at this time, use the PMC to perform processing as needed. If the main circuit overheats, an alarm occurs.
04	Open-phase detected in the converter main power supply	If an open-phase is detected in the main power supply, the warning signal is output. Since the spindle continues to operate at this time, use the PMC to perform processing as needed. About one minute (for the αiPS) or about five seconds (for the αiPSR) after the warning signal is output, an alarm occurs.
58	Converter main circuit overloaded	If the main circuit of the Power Supply (PS) is overloaded, the warning signal is output. Since the spindle continues to operate at this time, use the PMC to perform processing as needed. About one minute after the warning signal is output, an alarm occurs.
59	Converter cooling fan stopped	If the Power Supply (PS) cooling fan stops, the warning signal is output. Since the spindle continues to operate at this time, use the PMC to perform processing as needed. About one minute after the warning signal is output, an alarm occurs.
113	Converter radiator cooling fan stopped	If the Power Supply (PS) radiator cooling fan stops, the warning signal is output. Since the spindle continues to operate at this time, use the PMC to perform processing as needed. If the Power Supply (PS) main circuit overheats, an alarm occurs.
01	Motor overheat	When the motor temperature increases beyond the overheat warning detection level (set by a parameter), a warning signal is output. At this time, spindle operation is continued. So, perform necessary processing with the PMC. An alarm is issued when the motor temperature has reached the overheat alarm detection level.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn264	SPWRN8	SPWRN7	SPWRN6	SPWRN4	SPWRN4	SPWRN3	SPWRN2	SPWRN1
Fn265								SPWRN9

Diagnosis data

The status of a warning is displayed on the following diagnosis data.

712	Warning status of spindle

[Data type] Word spindle
The number of a warning caused on each spindle is indicated.
If there is no warning, 0 is indicated.

18.3 FAN MOTOR ABNORMALITY MONITORING FUNCTION AND COMMUNICATION RETRY MONITORING FUNCTION

18.3.1 Fan Motor Abnormality Monitoring Function

Overview

This function monitors the number of revolutions of the fan motor in the CNC control part to anticipate a critical fault. If the fan motor is defective, the function detects it as an alarm level or a warning level, and the alarm, the signal, and the warning are output.

> ⚠ **CAUTION**
> Exchange fan motors referring to the maintenance manual if the defect of the fan motor is detected. Even if the warning level is detected, CNC can be continuously operated. However, there is a possibility that fan motor is deteriorated. Exchange fan motors as soon as possible.

Specification

The CNC control unit fan motor operates as shown in Table 18.3.1 if it is defective.

Table 18.3.1 Operation of the PCB cooling fan when it is defective

State	Operation
When the warning level is detected.	Displaying the warning status Outputting the warning level detection signal WFAN
When the alarm level is detected.	Displaying the warning status Outputting the Alarm level detection signal SFAN Servo: Makes a single block stop. Spindle: Continues rotating.
	After 50 seconds Servo: Makes a feed hold stop. Spindle: Stops rotating. After 60 seconds Alarm 701 occurs. Note) System alarm 455 is generated for the CNC control unit where CPU card (high speed version) was installed about 60 seconds after the detection of the alarm level.

> **NOTE**
> When the alarm level is detected, the spindle stops rotating 50 seconds later, but it starts rotating again when the warning or alarm is cleared. To stop the rotation of the spindle completely, create a ladder that stops the spindle using alarm level detection signal SFAN.

Signal

Alarm level detection signal SFAN<Fn093.1>

[Classification] Output signal

[Operation] This signal is output when the alarm level is detected by fan motor in the CNC control unit.

[Function] Reports that the alarm level was detected by fan motor in the CNC control unit.

> **NOTE**
> It takes a maximum of five seconds after the detection of the alarm level until the alarm level detection signal SFAN is output.

Warning level detection signal WFAN<Fn093.3>

[Classification] Output signal

[Operation] This signal is output when the warning level is detected by fan motor in the CNC control unit.

[Function] Reports that the warning level was detected by fan motor in the CNC control unit.

> **NOTE**
> It takes a maximum of five seconds after the detection of the warning level until the warning level detection signal WFAN <Fn093.3> is output.

18.3.2 Communication Retry Monitoring Function

Overview

If the I/O Link retry counter value, Ethernet (embedded + fast), FL-net error detection count, and ECC (SRAM) correction counter exceed the parameter setting, signals are output to report a reduction in system reliability.

Specification

I/O Link retry counter value abnormality monitoring function

This function monitors the retry counter at 10 minute intervals, it outputs warning signals (WIOCH1, WIOCH2, WIOCH3) if the counter value is equal to or more than the setting of parameter No. 11530.

The threshold beyond which a warning occurs changes every eight hours depending on the energization time.

When the energization time is T (hour) and parameter No. 11530 is n:

$0 \le T \le 8$

A warning occurs when the retry counter value is equal to or greater than n.

$8 < T$

A warning occurs when the retry counter value is equal to or greater than $n \times T/8$ (rounded up).

The I/O Link retry counter makes a check at 10 minute intervals, so the warning signal is output at 10 minute intervals.

The counter value is accumulated until the power is turned off.

> **NOTE**
> The maximum value of the retry counter is 255.
> A warning is issued when an overflow occurs.

ECC correction counter value abnormality monitoring function

This function monitors the correction counter at 10 minute intervals, it outputs warning signal (WECCS<F0535.3>) if the counter value is equal to or more than the setting of parameter No. 11531.

The threshold beyond which a warning occurs changes every eight hours depending on the energization time.

When the energization time is T (hour) and parameter No. 11531 is n:

$0 \le T \le 8$

A warning occurs when the retry counter value is equal to or greater than n.

$8 < T$

A warning occurs when the retry counter value is equal to or greater than $n \times T/8$ (rounded up).

The ECC correction counter makes a check at 10 minute intervals, so the warning signal is output at 10 minute intervals.

The counter value is accumulated until the power is turned off.

> **NOTE**
> The maximum value of the correction counter is 4095.
> A warning is issued when an overflow occurs.

Ethernet error detection count abnormality monitoring function

- Error detection of embedded Ethernet
  Checks the error detection count of embedded Ethernet at 10 second intervals and, if the error detection count for the nearest one minute is equal to or greater than the setting of parameter No. 11532, this function outputs warning signal WETE<F0535.4>.

- Error detection of fast Ethernet
  Checks the error detection count of fast Ethernet at 10 second intervals and, if the error detection count for the nearest one minute is equal to or greater than the setting of parameter No. 11533, this function outputs warning signal WETE.

The error detection count is checked at 10 second intervals in both embedded Ethernet and fast Ethernet, so the warning signal is output at 10 second intervals. Turning off and back on the power resets the signal output.

> **NOTE**
> The maximum value for network error counters is 65535.
> If an overflow occurs, no warning is issued and the error count returns to 0.

FL-net error detection count abnormality monitoring function

Checks the error detection count of FL-net at 10 second intervals and, if the error detection count for the nearest one minute is equal to or greater than the setting of parameter No. 11534, this function outputs warning signal. FL-net1 and FL-net2 either use a set value of parameter No.11534. However, as for the warning signal, WFLN1 is output for FL-net1, and WFLN2 is output for FL-net2.

The error detection count is checked at 10 second intervals, so the warning signal is output at 10 second intervals. Turning off and back on the power resets the signal output.

In the FL-net/Ethernet coexisting function, checks total of the error detection count of FL-net and Ethernet at 10 second intervals and, if the error detection count for the nearest one minute is equal to or greater than the setting of parameter No. 11533, this function outputs warning signal WETF<F0535.5>.

> **NOTE**
> The maximum value for network error counters is 65535.
> If an overflow occurs, no warning is issued and the error count returns to 0.

Parameter

11530	Warning value (common to ch1, ch2, and ch3) for I/O link retry counter

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte
[Unit of data] Correction counter value/8 hours
[Valid data range] 0 to 127

Recommended setting = 0
Set a warning value for the I/O link communication retry counter.
When the value in the 8-hour error correction counter becomes higher than or equal to the setting, the warning signal is output.
If the setting is 0, monitoring is performed on a 5 occurrences/8 hours basis.

11531	Warning value for the ECC correction counter (SRAM)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte
[Unit of data] Correction counter value/8 hours
[Valid data range] 0 to 127

Recommended setting = 0
Set a warning value for the SRAM correction counter.
When the value in the 8-hour error correction counter becomes higher than or equal to the setting, the warning signal is output.
If the setting is 0, monitoring is performed on a 5 occurrences/8 hours basis.

11532	Warning value for the number of embedded-Ethernet error detection occurrences

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte
[Unit of data] Number of detection occurrences/minute
[Valid data range] 0 to 127

Recommended setting = 0
Set a warning value for the number of embedded Ethernet error detection occurrences.
When the number of 1- minute error detection occurrences becomes higher than or equal to the setting, the warning signal is output.
If the setting is 0, monitoring is performed on a 10 occurrences/minute basis.

11533	Warning value for the number of fast Ethernet error detection occurrences

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte
[Unit of data] Number of detection occurrences/minute
[Valid data range] 0 to 128
Recommended setting = 0
Set a warning value for the number of fast Ethernet error detection occurrences.
When the number of 1- minute error detection occurrences becomes higher than or equal to the setting, the warning signal is output.
If the setting is 0, monitoring is performed on a 10 occurrences/minute basis.

11534	Warning value (common to FL-net port 1 and 2) for the number of FL-net error detection occurrences

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte
[Unit of data] Number of detection occurrences/minute
[Valid data range] 0 to 127
Recommended setting = 0
Set a warning value for the number of FL-net error detection occurrences.
When the number of 1- minute error detection occurrences becomes higher than or equal to the setting, the warning signal is output.
If the setting is 0, monitoring is performed on a 3 occurrences/minute basis.

Signal

I/O Link 1 retry abnormality warning signal WIOCH1<F0535.0>

[Classification] Output signal
[Operation] This signal is output when a warning occurs.
[Function] In I/O Link, if an error is detected in received data, the data is transmitted again (retried) to prevent the system from going down due to a temporary communication error caused by noise or the like.
This signal is set to 1 when the data retry count per eight hours is equal to or greater than the value set in parameter No. 11530 (standard setting 0, which means five retries per eight hours) in the I/O link on the first channel side.
An I/O Link retry abnormality is monitored at 10 minute intervals, so this signal is output at 10 minute intervals. Once this signal is set to 1, the value is kept until the CNC power is turned off.

I/O Link 2 retry abnormality warning signal WIOCH2<F0535.1>

[Classification] Output signal
[Operation] This signal is output when a warning occurs.
[Function] This signal is set to 1 when the retry count per eight hours for I/O Link on the second channel side is equal to or greater than the setting of parameter No. 11530.
The other usage is the same as in I/O Link 1 retry failure warning signal WIOCH1.

I/O Link 3 retry abnormality warning signal WIOCH3<F0535.2>

[Classification] Output signal

[Operation] This signal is output when a warning occurs.

[Function] This signal is set to 1 when the retry count per eight hours for I/O Link on the third channel side is equal to or greater than the setting of parameter No. 11530.
The other usage is the same as in I/O Link 1 retry failure warning signal WIOCH1.

SRAM ECC abnormality warning signal WECCS<F0535.3>

[Classification] Output signal

[Operation] This signal is output when a warning occurs.

[Function] SRAM, which stores parameters, machining programs, offset data, etc., uses ECC (error correction code) to correct a single bit error. If the SRAM error correction count is equal to or greater than the setting of parameter No. 11531 (standard setting 0, which means five retries per eight hours), this signal is set to 1.
Once this signal is set to 1, the value is kept until the CNC power is turned off.

Embedded Ethernet communication abnormality warning signal WETE<F0535.4>

[Classification] Output signal

[Operation] This signal is output when a warning occurs.

[Function] If the total number of various errors detected by the communication hardware of the embedded Ethernet function is equal to or greater than the setting of parameter No. 11532 (standard setting 0, which means 10 retries per one minute), this signal is set to 1.
Once this signal is set to 1, the value is kept until the CNC power is turned off.

Fast Ethernet communication abnormality warning signal WETF<F0535.5>

[Classification] Output signal

[Operation] This signal is output when a warning occurs.

[Function] If the total number of various errors detected by the communication hardware of the fast Ethernet function is equal to or greater than the setting of parameter No. 11533 (standard setting 0, which means 10 retries per one minute), this signal is set to 1.
Once this signal is set to 1, the value is kept until the CNC power is turned off.

FL-net1 communication abnormality warning signal WFLN1<F0535.6>

[Classification] Output signal

[Operation] This signal is output when a warning occurs.

[Function] If the total number of various errors detected by the communication hardware of the FL-net1 function is equal to or greater than the setting of parameter No. 11534 (standard setting 0, which means 3 retries per one minute), this signal is set to 1.
Once this signal is set to 1, the value is kept until the CNC power is turned off.

FL-net2 communication abnormality warning signal WFLN2<F0535.7>

[Classification] Output signal

[Operation] This signal is output when a warning occurs.

[Function] If the total number of various errors detected by the communication hardware of the FL-net2 function is equal to or greater than the setting of parameter No. 11534 (standard setting 0, which means 3 retries per one minute), this signal is set to 1.
Once this signal is set to 1, the value is kept until the CNC power is turned off.

18.4 TROUBLE DIAGNOSIS

18.4.1 Outline

Investigating the cause of Servo/Spindle/CNC alarms becomes easier by diagnosis according to the guidance message of this function.
And when the thermal simulation data or disturbance level of servo axis exceeds the trouble forecast level, the machine can be safely stopped by detecting the breakdown beforehand by the use of the trouble forecast signal.
It introduces the example of the method of pursuing the cause of the alarm occurrence that uses the trouble diagnosis function as follows.

When alarm "SV0410 EXCESS ERROR (STOP)" is occurred.

It explains the procedure of the cause investigation when following alarm "SV0410 EXCESS ERROR (STOP)" is occurred.

Fig.18.4.1 (a) Alarm message screen (10.4-inch display unit)

The cause of occurrence is pursued by answering the question displayed on the screen according to the following procedures by soft key [YES] or [NO] though there are some occurrence factors of the alarm "SV0410 EXCESS ERROR (STOP)".

1 After pushing return soft key [<], press the continuous menu key until soft key [GUIDE]/[MONIT]/[W.GRPH] appears.

2 Press soft key [GUIDE].

Fig.18.4.1 (b) Soft key to select trouble diagnosis guidance screen (10.4-inch display unit)

Because the following screens are displayed, press soft key [YES].

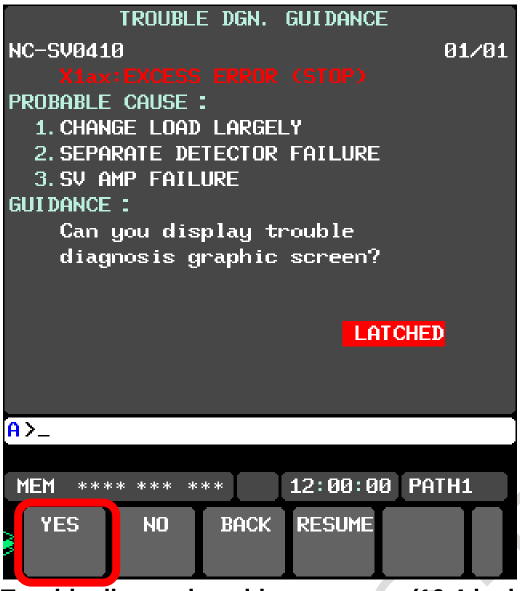


Fig.18.4.1 (c) Trouble diagnosis guidance screen (10.4-inch display unit)

The following guidances are displayed.

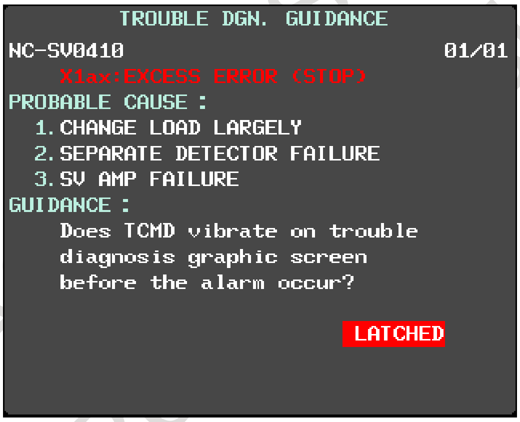


Fig.18.4.1 (d) Trouble diagnosis guidance screen (10.4-inch display unit)

Trouble diagnosis graphic screen is displayed according to the above guidance and the torque command is confirmed.

Trouble diagnosis graphic screen is displayed according to the following procedures.

1 Press return soft key [<].
2 Press soft key [W.GRPH].

Fig.18.4.1 (e) Soft key to select trouble diagnosis graphic screen (10.4-inch display unit)

3 Press soft key [W.PRM].

Fig.18.4.1 (f) Soft key to select trouble diagnosis parameter screen (10.4-inch display unit)

A necessary parameter for the graphic display is set as follows.

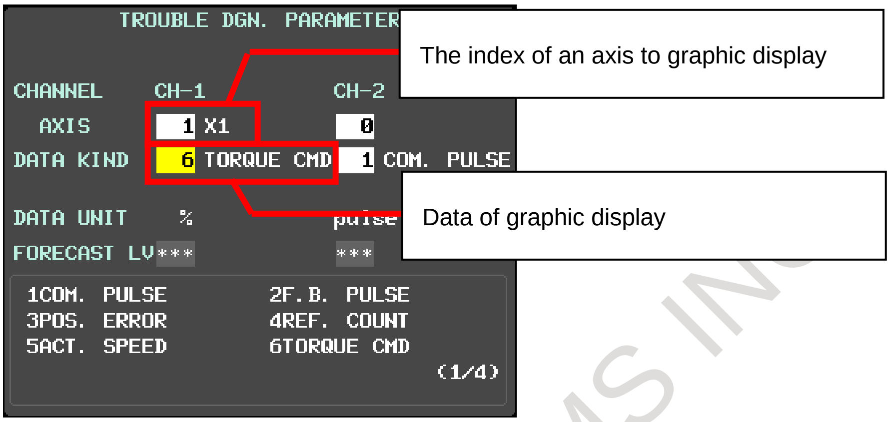


Fig.18.4.1 (g) Trouble diagnosis parameter screen (10.4-inch display unit)

The displayed axis is selected by inputting the index of an axis to the box next to "AXIS".
The displayed data kind is selected by inputting data kind number to the box next to "DATA KIND".

This time, "1" is set in the index of an axis to confirm the waveform of the torque command of the X1 axis and "6" is set to data kind number.

Trouble diagnosis graphic screen is displayed according to the following procedures.

1 Press return soft key [<].
2 Press soft key [G-ADJ.].

Fig.18.4.1 (h) Soft key to select trouble diagnosis graphic screen (10.4-inch display unit)

The graphical chart of the torque command immediately before occurring the alarm as follows is displayed.
The vertical line shown by the arrow on a right edge is time that the alarm is occurred.

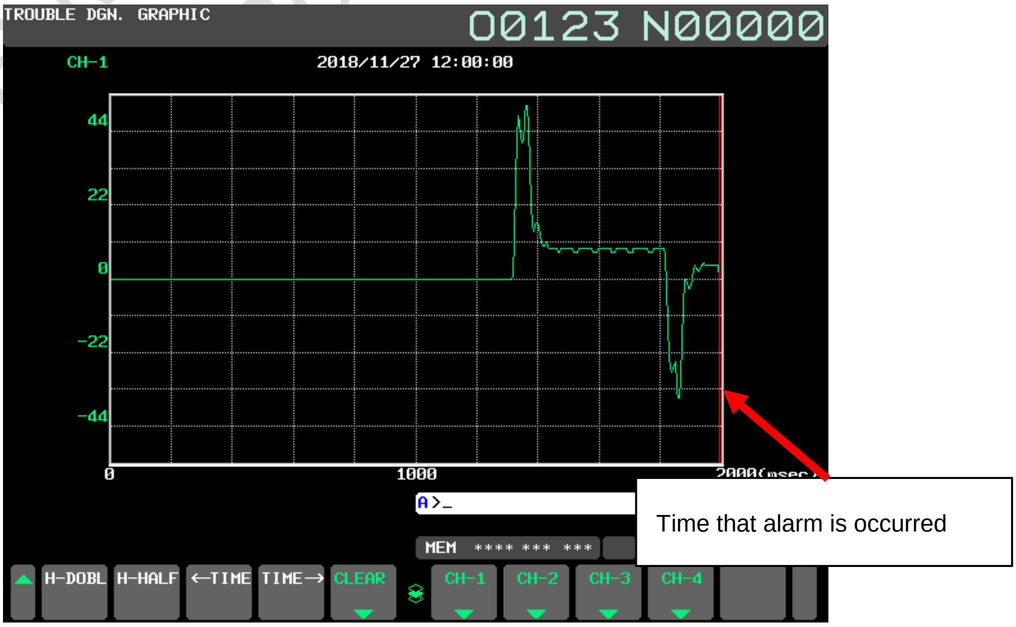


Fig.18.4.1 (i) Trouble diagnosis graphic screen (10.4-inch display unit)

Return to trouble diagnosis guidance screen, and press the soft key [NO], because it can be judged that the torque command doesn't vibrate from this graphical chart.
Trouble diagnosis guidance screen is displayed according to the following procedures.

1. Press return soft key [<].
2. Press soft key [GUIDE].

Fig.18.4.1 (j) Soft key to select trouble diagnosis guidance screen (10.4-inch display unit)

Press soft key [NO] after pressing the soft key [(OPRT)] on the trouble diagnosis guidance screen.

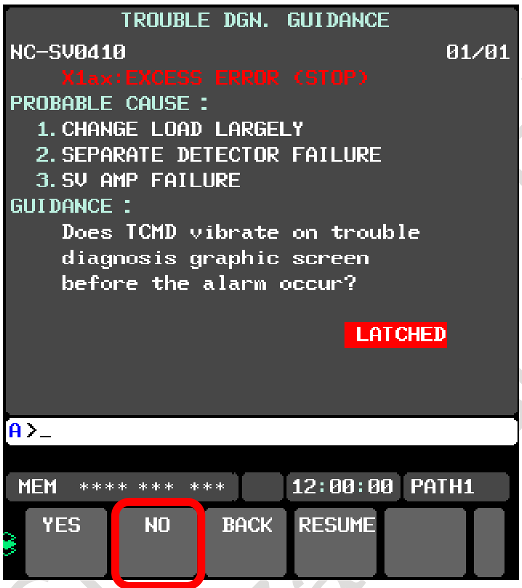


Fig.18.4.1 (k) Trouble diagnosis guidance screen (10.4-inch display unit)

When soft key [NO] is pressed, the following guidances are displayed.

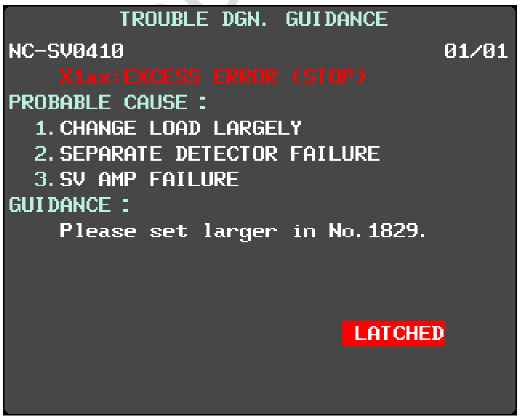


Fig.18.4.1 (l) Trouble diagnosis guidance screen (10.4-inch display unit)

The cause of the alarm is specified like this, and cope according to the content of the guidance.

18.4.2 Investigation procedure of cause of alarm occurrence

The cause of occurrence can be investigated according to the following procedures when Servo/Spindle alarms are occurred.

> **NOTE**
> Status display of the trouble diagnosis screen should be [SAMPLING] to diagnose it when the the servo alarm/spindle alarm is occurred.
> Change in the state of [SAMPLING] surely referring to "Clear of the preserved data" in Chapter 18.4.4 when status display is not [SAMPLING].

Investigation procedure (for 8.4/10.4-inch display unit)

Procedure

1. Press function key , and the message screen is displayed.
2. After pushing return soft key [<], press the continuous menu key  until soft key [GUIDE]/[MONIT]/[W.GRPH] appears.
3. Press soft key [GUIDE].
   The trouble diagnosis guidance screen is displayed.
   The guidance to specify the cause of the occurred alarm is displayed on the trouble diagnosis guidance screen.
   The cause of occurrence and the coping method of the alarm are displayed by answering in soft key [YES]/[NO] referring to servo/spindle monitor information on the trouble diagnosis monitor screen, the waveform data of the trouble diagnosis graphic screen, and the diagnosis screen, etc.

Investigation procedure (for 15/19-inch display unit)

Procedure

1. Press function key , and the message screen is displayed.
2. Press the vertical soft key [NEXT PAGE] until vertical soft key [GUIDE]/[MONIT]/[W.GRPH] appears.
3. Press the vertical soft key [GUIDE].
   The trouble diagnosis guidance screen is displayed.
   The guidance to specify the cause of the occurred alarm is displayed on the trouble diagnosis guidance screen.
   The cause of occurrence and the coping method of the alarm are displayed by answering in horizontal soft key [YES]/[NO] referring to servo/spindle monitor information on the trouble diagnosis monitor screen, the waveform data of the trouble diagnosis graphic screen, and the diagnosis screen, etc.

18.4.3 Trouble diagnosis guidance screen

Outline

The trouble diagnosis guidance screen displays the guidance message to specify the cause of an alarm. The cause of the alarm can be specified by confirmation/coping according to the guidance, and answering by soft key [YES]/[NO] in a short time.

Display procedure of trouble diagnosis guidance screen (for 8.4/10.4-inch display unit)

Procedure

1. Press function key .
2. Press the continuous menu key  until soft key [GUIDE] appears.
3. Press soft key [GUIDE]. The trouble diagnosis guidance screen is displayed as follows.

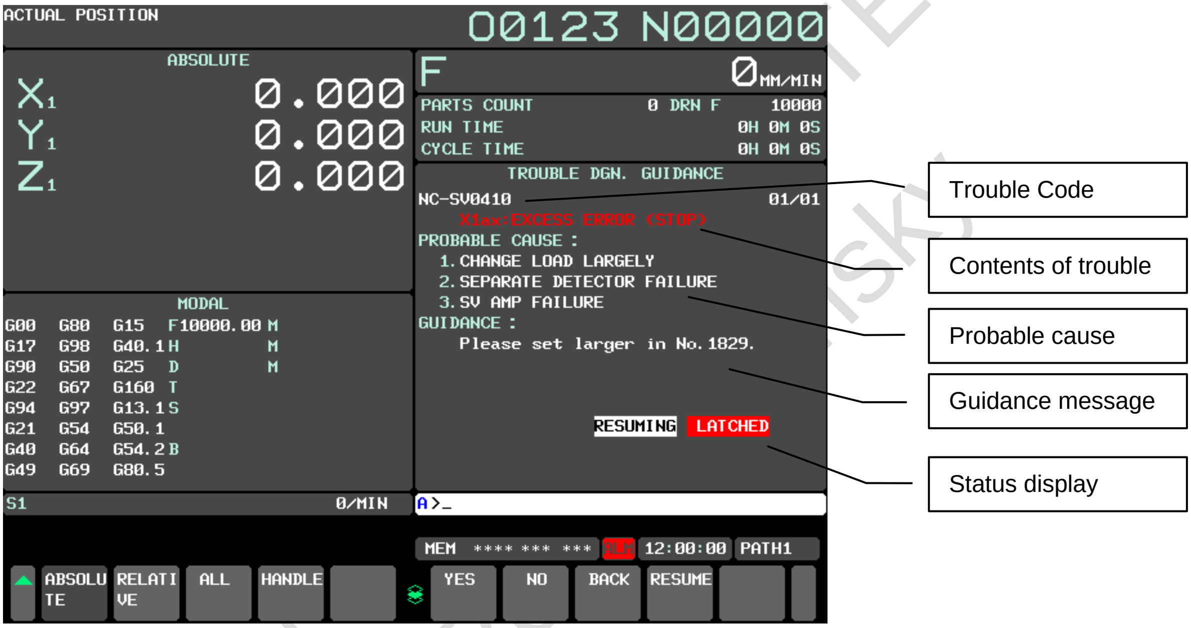


Fig.18.4.3 (a) Trouble diagnosis guidance screen (10.4-inch display unit)

Display procedure of trouble diagnosis guidance screen (for 15/19-inch display unit)

Procedure

1. Press function key .
2. Press the vertical soft key [NEXT PAGE] until vertical soft key [GUIDE] appears.
3. Press the vertical soft key [GUIDE]. The trouble diagnosis guidance screen is displayed.

Explanation

- Contents of Display

Trouble Code : "NC"-"Alarm type + alarm number" is displayed as code for identifying alarm.
Contents of Trouble : Alarm Message.
Probable Cause : Probable Cause of alarm.
Guidance message : Question to find the cause or answer to remove trouble is displayed.
Status display :

[RESUMING] :

When press soft key [RESUME] , the guidance message which was displayed before CNC power turned off is displayed again. Then [RESUMING] appears.

[SAMPLING] / [LATCHED] :

In CNC, there are two states in the state that the state (sampling state) to sample servo/spindle data whenever it usually operates and servo/spindle alarm are occurred and the sampling stops (preserved state). [SAMPLING] or [LATCHED] display show the two states.

The sampling of servo/spindle monitor data is stopped when the servo/spindle alarm is occurred, and the data when the alarm is occurred is kept in CNC.

It is automatically diagnosed that it switches to the trouble diagnosis guidance screen in [LATCHED] state after the alarm is occurred, and the diagnosis to servo/spindle alarm is displayed partially in CNC based on servo/spindle data when the alarm is occurred.

There is a possibility that a correct diagnosis cannot be done when the alarm number diagnosed on the trouble diagnosis guidance screen and the alarm number when monitor data was preserved when the alarm is occurred are different when the alarm is occurred next if the sampling is not begun. In this case, the display of [LATCHED] state blinks.

Clear the kept data on the screen of either the trouble diagnosis guidance screen, the trouble diagnosis monitor screen, or the trouble diagnosis graphic screen to begin the sampling again in preparation for the alarm occurred next after status display changes into [LATCHED] state once. after diagnosing the cause of occurrence and the corrective action of the alarm.

When the kept data is cleared, [LATCHED] state changes into [SAMPLING] state, and restarts the sampling of servo/spindle monitor data.

Operation

Press the soft key [(OPRT)] on the trouble diagnosis guidance screen. (Only for 8.4/10.4-inch display unit)

- Change of Guidance

Soft key [YES]/[NO] :

Check contents of guidance message, and, answer by pressing soft key [YES] or [NO]. Then the next guidance message is displayed.

In some cases CNC automatically checks and judges contents of guidance. In this case the next guidance message is automatically displayed.

Automatic diagnosis is not done in case that CNC power turns off once after servo /spindle monitor data is kept.

Soft key [BACK] :

Guidance message returns back 1 step.

It is possible to trace back the guidance message when soft key [YES]/[NO] is pressed by mistake.

Soft key [RESUME] :

Guidance message which was displayed before CNC power turned off is displayed again.

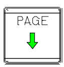
The diagnosis can be continued from the guidance displayed again.

However, it is not possible to go back to the step before the point where soft key [BACK] is pressed.

Status display “RESUMING” is displayed during the guidance that starts by pressing soft key [RESUME].

- Change of alarm

When several alarms occur in same time, pressing page key 

/ 

can select the guidance message.

18.4.4 Trouble diagnosis monitor screen

Outline

Trouble diagnosis monitor screen displays information when the alarm is occurred necessary to diagnose the servo/spindle alarm.

Servo/spindle monitor data on "1-4 immediately before occurrence of the alarm" for four samplings immediately before and "The latest alarm is occurred" can be displayed as data when the alarm is occurred.
Moreover, "Present" of servo/spindle monitor data when usually operating can be displayed.

Display procedure of trouble diagnosis monitor screen (for 8.4/10.4-inch display unit)

Procedure

1 Press function key [MESSAGE].

2 Press the continuous menu key [▷] until soft key [MONIT] appears.

3 Press soft key [MONIT].

The following screen is sample of displaying data of X axis (servo) when the alarm occurred.

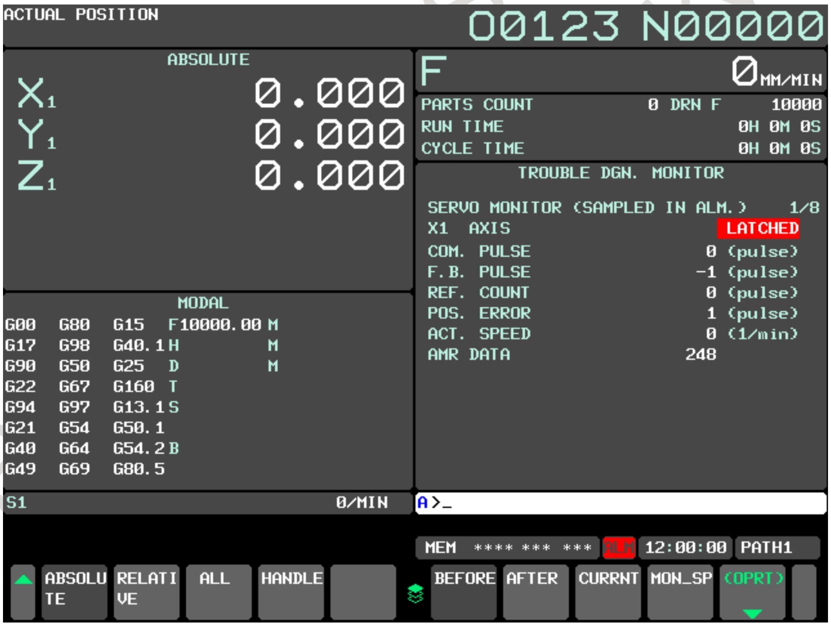


Fig.18.4.4 (a) Trouble diagnosis monitor screen (10.4-inch display unit)

4 The servo monitor data screen and the spindle monitor data screen can be switched by pushing soft key [MON_SP] / [MON_SV].

Display procedure of trouble diagnosis monitor screen (for 15/19-inch display unit)

Procedure

1 Press function key [MESSAGE].

2 Press the vertical soft key [NEXT PAGE] until vertical soft key [MONIT] appears.

3 Press the vertical soft key [MONIT].

4 The servo monitor data screen and the spindle monitor data screen can be switched by pushing the vertical soft key [MON_SP] / [MON_SV].

Explanation

- Data displayed in monitor Screen

Displayed data in monitor screen is showed below.

Table 18.4.4 (a) Data of servo motor

Data (Unit)	Data type	Display range	Data explanation	Page number
COMMAND PULSE (pulse)	2 Word	±99999999		1
F.B. PULSE (pulse)	2 Word	±99999999	(NOTE 3)	1
REF.COUNTER (pulse)	2 Word	±99999999		1
POS. ERROR (pulse)	2 Word	±99999999		1
ACTUAL SPEED (1/min)	Word	-32768 to +32767		1
AMR DATA	Word	0 to +255	Data of pole position of the rotor “256” shows 360 degree as electrical angle	1
TORQUE COMMAND (%)	Word	±400	Torque command. “100” shows maximum torque	2
EFFECTIVE CURRENT (%)	Word	±400	“100” shows maximum current of amplifier	2
MOTOR CURRENT (A)	Word	±4096		2
DISTURBANCE LEVEL (%)	Word	0 to +200	“100” shows the alarm level of Unexpected disturbance torque detection function	2
HEAT SIMULATION (%)	Word	0 to +100	“100” shows the alarm level of OVC alarm.	2
ARBITRARY DATA1	Word	-32768 to +32767	(NOTE 2)	2
ARBITRARY DATA2	Word	-32768 to +32767	(NOTE 2)	2
DC LINK VOLT. (V)	Word	0 to +1023	Instantaneous value (NOTE 3)	3 (NOTE 6)
PS VOLTAGE RMS (Vrms)	Word	0 to +1023	Average value during 1 cycle of power source (NOTE 3)	3 (NOTE 6)
PS VOLT.UMBALANCE (%)	Word	0 to +15.5	Average value during 1 cycle of power source (NOTE 3)	3 (NOTE 6)
PS VOLTAGE THD (%)	Word	0 to +31.5	Average value of THD(Total Harmonic Distortion) during 1 cycle of power frequency (NOTE 3)	3 (NOTE 6)
PS CURRENT (A)	Word	0 to +2047	Average value of current amplitude during 1 cycle of power source (NOTE 3)	3 (NOTE 6)
PS STATUS FLAG 1	Bit	0 / 1	(NOTE 4)	3 (NOTE 6)
PS STATUS FLAG 2	Bit	0 / 1	(NOTE 4)	3 (NOTE 6)
PS STATUS FLAG 3	Bit	0 / 1	(NOTE 4)	3 (NOTE 6)
PS STATUS FLAG 4	Bit	0 / 1	(NOTE 4)	3 (NOTE 6)
PS INPUT FREQ (Hz)	Word	0 to +102.3	Average value during 1 cycle of power source (NOTE 3)	3 (NOTE 6)
SV INS.INFO.	Byte	0 to 3F	Various status flags at using Leakage detection function	4 (NOTE 6)
SV INS.RESISTANCE (MΩ)	Word	0 to +102.1	Value of insulation resistance at using Leakage detection function (NOTE 3)	4 (NOTE 6)
PS INT.TMP. (℃)	Byte	0 to +255	(NOTE 3)	5 (NOTE 6)
PS HEAT SINK TMP. (℃)	Byte	0 to +255	(NOTE 3)	5 (NOTE 6)

Data (Unit)	Data type	Display range	Data explanation	Page number
SV INT.TMP. (℃)	Byte	0 to +255	(NOTE 3)	5 (NOTE 6)
SV HEAT SINK TMP. (℃)	Byte	0 to +255	(NOTE 3)	5 (NOTE 6)
AMP GROUP/SLAVE	Byte	0 to +255	Group number and Slave number of power supply (PS)	5 (NOTE 6)
PS DGN.INFO.	Word	0 to FFFF	Status flags of power supply(PS)	5 (NOTE 6)
AMP COMM.ERR.INF.	Word	0 to FFFF	Error information flags for communication between amplifiers	5 (NOTE 6)
SV DGN.INFO.	Word	0 to FFFF	Diagnosis flags for servo amplifier(SV)	5 (NOTE 6)
SV FSSB UPR.ERR.	2Word	0 to FFFF	(NOTE 5)	6 (NOTE 6)
SV FSSB LWR.ERR.	2Word	0 to FFFF	(NOTE 5)	6 (NOTE 6)
SV FSSB UPR.JTR.	2Word	0 to FFFF	(NOTE 5)	6 (NOTE 6)
SV FSSB LWR.JTR.	2Word	0 to FFFF	(NOTE 5)	6 (NOTE 6)
SDU FSSB UPR.ERR.	2Word	0 to FFFF	(NOTE 5)	6 (NOTE 6)
SDU FSSB LWR.ERR.	2Word	0 to FFFF	(NOTE 5)	6 (NOTE 6)
SDU FSSB UPR.JTR.	2Word	0 to FFFF	(NOTE 5)	6 (NOTE 6)
SDU FSSB LWR.JTR.	2Word	0 to FFFF	(NOTE 5)	6 (NOTE 6)
INT.DTCT.INTP.CNT	Word	0 to +65535	The number of data interpolation when the feedback data of internal detector is disturbed by noise	7 (NOTE 7)
INT.DTCT.COM.CNT	Word	0 to +65535	The number of communication error when the feedback data of internal detector is disturbed by noise	7 (NOTE 7)
INT.DETECTOR WRN.	Word	0 to FFFF	Warning information of internal detector	7 (NOTE 6)
EXT.DTCT.INTP.CNT	Word	0 to +65535	The number of data interpolation when the feedback data of external detector is disturbed by noise	7 (NOTE 7)
EXT.DTCT.COM.CNT	Word	0 to +65535	The number of communication error when the feedback data of external detector is disturbed by noise	7 (NOTE 7)
EXT.DETECTOR WRN.	Word	0 to FFFF	Warning information of external detector	7 (NOTE 6)
SV DATA 1	Word	-32768 to +32767	(Note2)	7 (NOTE 6)
SV DATA 2	Word	-32768 to +32767	(Note2)	7 (NOTE 6)
SV DATA 3	Word	-32768 to +32767	(Note2)	7 (NOTE 6)
SV DATA 4	Word	-32768 to +32767	(Note2)	7 (NOTE 6)

NOTE

1 The data range shows the range in which a data can be monitored in this function. It doesn't show the system efficiency or rated value.
2 ARBITRARY DATA1,2 and SV DATA1-4 are used for service operation by FANUC.
3 The displayed values as Voltage, Current, Frequency, Resistance, and Temperature are approximate value and they have some error. If you want to know the precise values, measure them by exclusive equipments.
4 PS STATUS FLAG1-4 are used for service operation by FANUC.
5 FSSB ERROR and FSSB JITTER show the status of FSSB communication. They are used for service operation by FANUC.
6 These items are not displayed in the system of G321 and 31 G331.
7 These items are displayed on the 3 page in the system of G321 and G331.

Table 18.4.4 (b) Data of spindle motor

Data (Unit)	Data type	Display range	Data explanation	Page number
OPERATION	Character	*****		1
GEAR/OUT SEL	Character	*****		1
COMMAND PULSE(pulse)	2Word	±99999999		1
COMMAND SPEED(1/min)	Word	-32768 to +32767	(NOTE 2)	1
CONTROL INPUT	Character	*****		1
CONTROL OUTPUT	Character	*****		1
ACT.SPINDLE SPEED (1/min)	Word	-32768 to +32767		2
ACT.MOTOR SPEED (1/min)	Word	-32768 to +32767	(NOTE 2)	2
LOAD METER (%)	Word	0 to +10000		2
TORQUE CMD(%)	Word	-100 to +100	Torque command. "100" shows the maximum torque.	2 (NOTE 7)
MOTOR CURRENT (A)	Word	-4096 to +4095		2 (NOTE 7)
HEAT SIMU.(MOTOR) (%)	Byte	0 to +100	"100" shows the alarm level of OVC alarm.	2 (NOTE 7)
HEAT SIMU.(AMP) (%)	Byte	0 to +100	"100" shows the alarm level of OVC alarm.	2 (NOTE 7)
POS. ERROR (pulse)	2Word	±99999999		2
SYN ERR (pulse)	2Word	±99999999		2
DC LINK VOLT. (V)	Word	0 to +1023	Instantaneous value (NOTE 3)	3 (NOTE 7)
PS VOLTAGE RMS (Vrms)	Word	0 to +1023	Average value during 1 cycle of power source (NOTE 3)	3 (NOTE 7)
PS VOLT.UMBALANCE (%)	Word	0 to +15.5	Average value during 1 cycle of power source (NOTE 3)	3 (NOTE 7)
PS VOLTAGE THD (%)	Word	0 to +31.5	Average value of THD(Total Harmonic Distortion) during 1 cycle of power source (NOTE 3)	3 (NOTE 7)
PS CURRENT (A)	Word	0 to +1023	Average value of current amplitude during 1 cycle of power source (NOTE 3)	3 (NOTE 7)
PS STATUS FLAG 1	Bit	0 / 1	(NOTE 5)	3 (NOTE 7)
PS STATUS FLAG 2	Bit	0 / 1	(NOTE 5)	3 (NOTE 7)
PS STATUS FLAG 3	Bit	0 / 1	(NOTE 5)	3 (NOTE 7)

Data (Unit)	Data type	Display range	Data explanation	Page number
PS STATUS FLAG 4	Bit	0 / 1	(NOTE 5)	3 (NOTE 7)
PS INPUT FREQ (Hz)	Word	0 to +102.3	Average value during 1 cycle of power source (NOTE 3)	3 (NOTE 7)
SP INS.INFO.	Byte	0 to 3F	Various status flags at using Leakage detection function (NOTE 3)	4 (NOTE 7)
SP INS.RESISTANCE (MΩ)	Word	0 to +102.1	Value of insulation resistance at using Leakage detection function (NOTE 3)	4 (NOTE 7)
PS INT.TMP. (℃)	Byte	0 to +255	(NOTE 3)	5 (NOTE 7)
PS HEAT SINK TMP. (℃)	Byte	0 to +255	(NOTE 3)	5 (NOTE 7)
SP INT.TMP. (℃)	Byte	0 to +255	(NOTE 3)	5 (NOTE 7)
SP HEAT SINK TMP. (℃)	Byte	0 to +255	(NOTE 3)	5 (NOTE 7)
AMP GROUP/SLAVE	Byte	0 to +255	Group number and Slave number of power supply (PS)	5 (NOTE 7)
PS DGN.INFO.	Word	0 to FFFF	Status flags of power supply(PS)	5 (NOTE 7)
AMP COMM.ERR.INF.	Word	0 to FFFF	Error information flags for communication between amplifiers	5 (NOTE 7)
SP DGN.INFO.	Word	0 to FFFF	Diagnosis flags for spindle amplifier(SP)	5 (NOTE 7)
SP FSSB UPR.ERR.	2Word	0 to FFFF	(NOTE 6)	6 (NOTE 7)
SP FSSB LWR.ERR.	2Word	0 to FFFF	(NOTE 6)	6 (NOTE 7)
SP FSSB UPR.JTR.	2Word	0 to FFFF	(NOTE 6)	6 (NOTE 7)
SP FSSB LWR.JTR.	2Word	0 to FFFF	(NOTE 6)	6 (NOTE 7)
INT.A/B AMPLITUDE (V)	Byte	0 to +2.55	Amplitude of internal analog sensor	7 (NOTE 7)
INT.A/B MAX FLUCT (mV)	Byte	0 to +1270	Maximum fluctuation ratio of internal analog sensor	7 (NOTE 7)
INT.A/B OFFSET A (mV)	Byte	-1280 to +1270	Offset of A-phase of internal analog sensor	7 (NOTE 7)
INT.A/B OFFSET B (mV)	Byte	-1280 to +1270	Offset of B-phase of internal analog sensor	7 (NOTE 7)
INT.A/B NOISE CNT	Word	-32768 to +32767	The number of noise when the feedback data of internal analog sensor is disturbed	7 (NOTE 7)
EXT.A/B AMPLITUDE (V)	Byte	0 to +2.55	Amplitude of external analog sensor	7 (NOTE 7)
EXT.A/B MAX FLUCT (mV)	Byte	0 to +1270	Maximum fluctuation ratio of external analog sensor	7 (NOTE 7)
EXT.A/B OFFSET A (mV)	Byte	-1280 to +1270	Offset of A-phase of external analog sensor	7 (NOTE 7)
EXT.A/B OFFSET B mV)	Byte	-1280 to +1270	Offset of B-phase of external analog sensor	7 (NOTE 7)
EXT.A/B NOISE CNT	Word	-32768 to +32767	The number of noise when the feedback data of external analog sensor is disturbed	7 (NOTE 7)
INT.SRAL INTP.CNT	Word	0 to +65535	The number of data interpolation when the feedback data of internal serial sensor is disturbed by noise	8 (NOTE 7)

Data (Unit)	Data type	Display range	Data explanation	Page number
INT.SRAL COM.CNT	Word	0 to +65535	The number of communication error when the feedback data of internal serial sensor is disturbed by noise	8 (NOTE 7)
INT.SRAL WRN.	Word	0 to FFFF	Warning information of internal serial sensor	8 (NOTE 7)
EXT.SRAL INTP.CNT	Word	0 to +65535	The number of data interpolation when the feedback data of external serial sensor is disturbed by noise	8 (NOTE 7)
EXT.SRAL COM.CNT	Word	0 to +65535	The number of communication error when the feedback data of external serial sensor is disturbed by noise	8 (NOTE 7)
EXT.SRAL WRN.	Word	0 to FFFF	Warning information of external serial sensor	8 (NOTE 7)
SP DATA 1	Word	-32768 to +32767	(NOTE 4)	8 (NOTE 7)
SP DATA 2	Word	-32768 to +32767	(NOTE 4)	8 (NOTE 7)
SP DATA 3	Word	-32768 to +32767	(NOTE 4)	8 (NOTE 7)
SP DATA 4	Word	-32768 to +32767	(NOTE 4)	8 (NOTE 7)

NOTE

1 The data range shows the range in which a data can be monitored in this function. It doesn't show the system efficiency or rated value.

2 To display COMMAND SPEED and ACTUAL MOTOR SPEED, it's necessary to set following parameters.
- No.4020 : Maximum motor speed of main spindle
- No.4196 : Maximum motor speed of sub spindle (in case of using Spindle switching control)

3 The displayed values as Voltage, Current, Frequency, Resistance, and Temperature are approximate value and they have some error. If you want to know the precise values, measure them by exclusive equipments.

4 SP DATA1-4 are used for service operation by FANUC

5 PS STATUS FLAG1-4 are used for service operation by FANUC.

6 FSSB ERROR and FSSB JITTER show the status of FSSB communication. They are used for service operation by FANUC.

7 These items are not displayed in the system of G321 and G331.

Table 18.4.4 (c) Data of others

Data	Data form	Page number
The latest latch date	Year (Two last digits) / Month (Two digits) / Day (Two digits)	8 (servo), 9 (spindle)
The latest latch time	Hour (Two digits) / Minute (Two digits) / Second (Two digits)	
Executed file name at latched	Character	
Executed N number at latched	N+ number of five digits	

Operation

- Servo monitor data screen

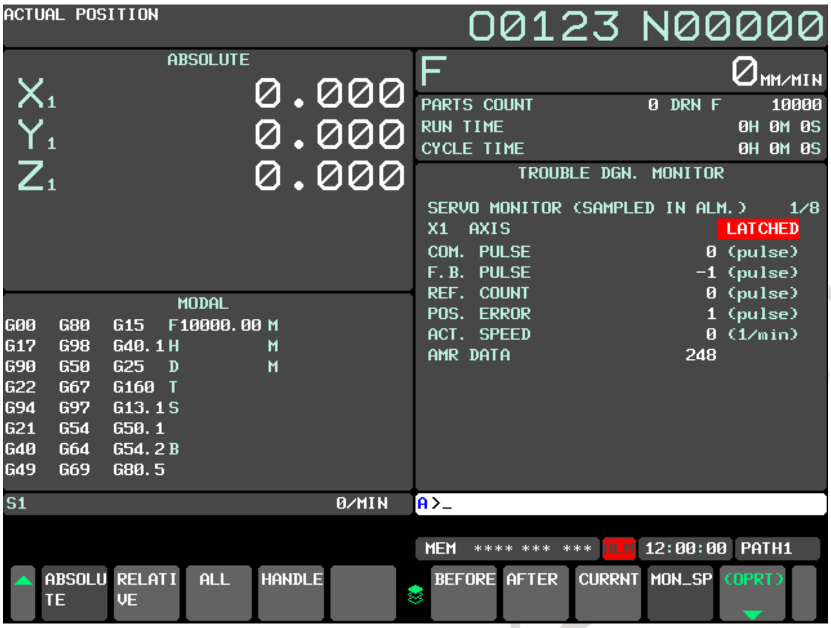
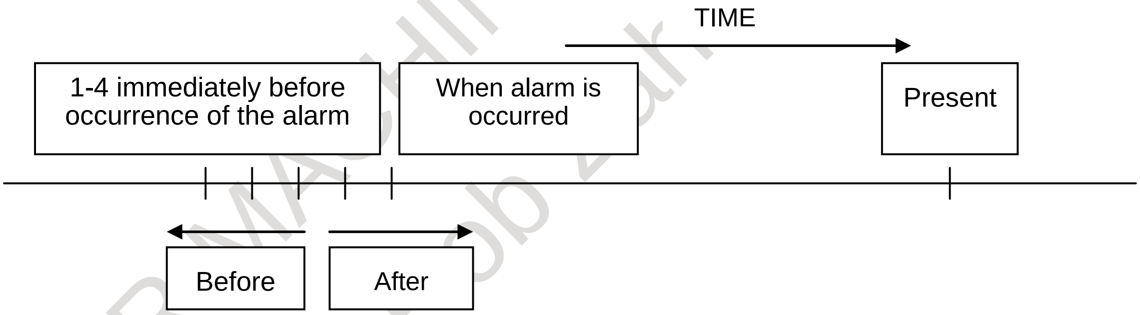


Fig.18.4.4 (b) Trouble diagnosis monitor screen (Servo monitor data) (10.4-inch display unit)

Servo monitor data can be selected by pressing soft key [NEW]/[CURRNT]/[BEFORE]/[AFTER].
Soft key [NEW] and [CURRNT] is displayed alternately.
Moreover, Soft key [BEFORE]/[AFTER] is displayed to press soft key [NEW], and if soft key [CURRNT] is pressed, it is not displayed.



Soft key [NEW]	: Servo monitor data when the alarm is occurred is displayed.
Soft key [CURRNT]	: Present servo monitor data is displayed.
Soft key [BEFORE]	: When the alarm is occurred and status display changes into [LATCHED] state, data before four samplings immediately before occurrence of the alarm is preserved in CNC. Preserved servo monitor data changes and is displayed by pressing this soft key as "SAMPLED IN ALM"→"BEFORE ALM 1"→"BEFORE ALM 2"→"BEFORE ALM 3"→"BEFORE ALM 4".
Soft key [AFTER]	: As well as the above operation, Preserved servo monitor data changes and is displayed by pressing this soft key as "BEFORE ALM 4"→"BEFORE ALM 3"→"BEFORE ALM 2"→"BEFORE ALM 1"→"SAMPLED IN ALM".
Soft key [MON_SP]	: Spindle monitor data screen is displayed.

NOTE

Monitor data of 1 - 4 samplings immediately before occurrence of the alarm is not displayed since 4 pages.

- Spindle monitor data screen

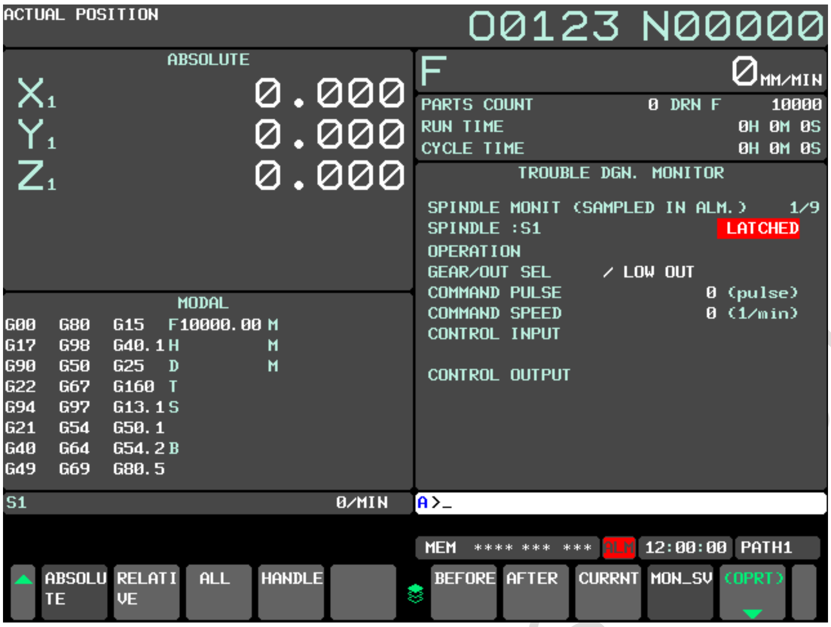
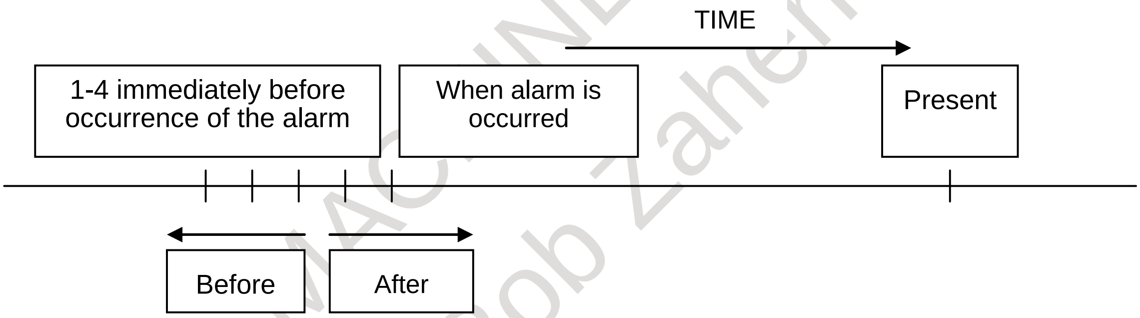


Fig.18.4.4 (c) Trouble diagnosis monitor screen (Spindle monitor data) (10.4-inch display unit)

Spindle monitor data can be selected by pressing soft key [NEW]/[CURRNT]/[BEFORE]/[AFTER].
Soft key [NEW] and [CURRNT] is displayed alternately.
Moreover, Soft key [BEFORE]/[AFTER] is displayed to press soft key [NEW], and if soft key [CURRNT] is pressed, it is not displayed.



Soft key [NEW]	: Spindle monitor data when the alarm is occurred is displayed.
Soft key [CURRNT]	: Present Spindle monitor data is displayed.
Soft key [BEFORE]	: When the alarm is occurred and status display changes into [LATCHED] state, data before four samplings immediately before occurrence of the alarm is preserved in CNC. Preserved Spindle monitor data changes and is displayed by pressing this soft key as "SAMPLED IN ALM"→"BEFORE ALM 1"→"BEFORE ALM 2"→"BEFORE ALM 3"→"BEFORE ALM 4".
Soft key [AFTER]	: As well as the above operation, Preserved Spindle monitor data changes and is displayed by pressing this soft key as "BEFORE ALM 4"→"BEFORE ALM 3"→"BEFORE ALM 2"→"BEFORE ALM 1"→"SAMPLED IN ALM".
Soft key [MON_SV]	: Servo monitor data screen is displayed.

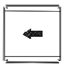
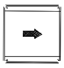
NOTE

Spindle monitor data that can the above operation is only an item from page 2 to page 3.
The value is displayed in the item on page 1 and since 4 pages of the spindle monitor data screen the value at present when status display is [SAMPLING], or the value at the alarm is occurred when status display is [LATCHED].

- Change of displayed page

Displayed page is switched by pressing page key  / .

- Change of displayed axis

Displayed axis is switched by pressing cursor key  / .

- Preservation of data when alarm is occurred

When status display is in the state of [SAMPLING], and servo alarm/spindle alarm is occurred newly, "Data at the latest alarm occurrence" and "Data of 1-4 immediately before occurrence of the alarm" of the alarm occurrence axis are preserved.
As a result, [SAMPLING] of status display on the right of the screen changes into [LATCHED].

> **NOTE**
> As for both servos and spindles, when the alarm as two or more axes at the same time is occurred, only the data of the axis of the least index of an axis is preserved.

- Clear of preserved data

After releasing all servo/spindle alarm, clear "Data at the latest alarm occurrence" and "Data of 1-4 immediately before occurrence of the alarm".
As a result, [LATCHED] of status display on the right of the screen changes into [SAMPLING]. Moreover, the waveform data (various data of each axis) displayed on the trouble diagnosis graphic screen is cleared at the same time.

Clear operation procedure of preserved data (for 8.4/10.4-inch display unit)

Procedure

1 The soft key display changes as follows if soft key [(OPER)] is pressed on the trouble diagnosis monitor screen, the trouble diagnosis graphic screen or the trouble diagnosis parameter screen.
(The below figure is a trouble diagnosis monitor screen.)

Fig.18.4.4 (d) Trouble diagnosis monitor screen (10.4-inch display unit)

2 Press the soft key [CLEAR].
The soft key display changes as follows.
Press the soft key [CAN] when you cancel the clear operation.

Fig.18.4.4 (e) Trouble diagnosis monitor screen (10.4-inch display unit)

3 Press the soft key [EXEC].
[LATCHED] of status display on the right of the screen changes into [SAMPLING]

Fig.18.4.4 (f) Trouble diagnosis monitor screen (10.4-inch display unit)

Clear operation procedure of preserved data (for 15/19-inch display unit)

Procedure

1. The horizontal soft key [CLEAR] deletion is displayed on the trouble diagnosis monitor screen, the trouble diagnosis graphic screen or the trouble diagnosis parameter screen.
2. Press the horizontal soft key [CLEAR].
   Press the horizontal soft key [CAN] when you cancel the clear operation.
3. Press the horizontal soft key [EXEC].
   [LATCHED] of status display on the right of the screen changes into [SAMPLING]

18.4.5 Trouble diagnosis graphic screen

Outline

The waveform data immediately before occurrence of the alarm necessary for the diagnosis when the servo alarm / spindle alarm is occurred is displayed.

Trouble diagnosis graphic screen

The servo data / spindle data for a few seconds immediately before occurrence of the alarm is displayed by waveform on the screen. The data of maximum 4 kinds can be displayed at the same time.
The displayed data is set on the trouble diagnosis parameter screen.

Display procedure of trouble diagnosis graphic screen (for 8.4/10.4-inch display unit)

Procedure

1. Press function key ? MESSAGE.
2. Press the continuous menu key until soft key [W.GRPH] appears.
3. Press soft key [W.GRPH].
4. Press soft key [G-ADJ.].

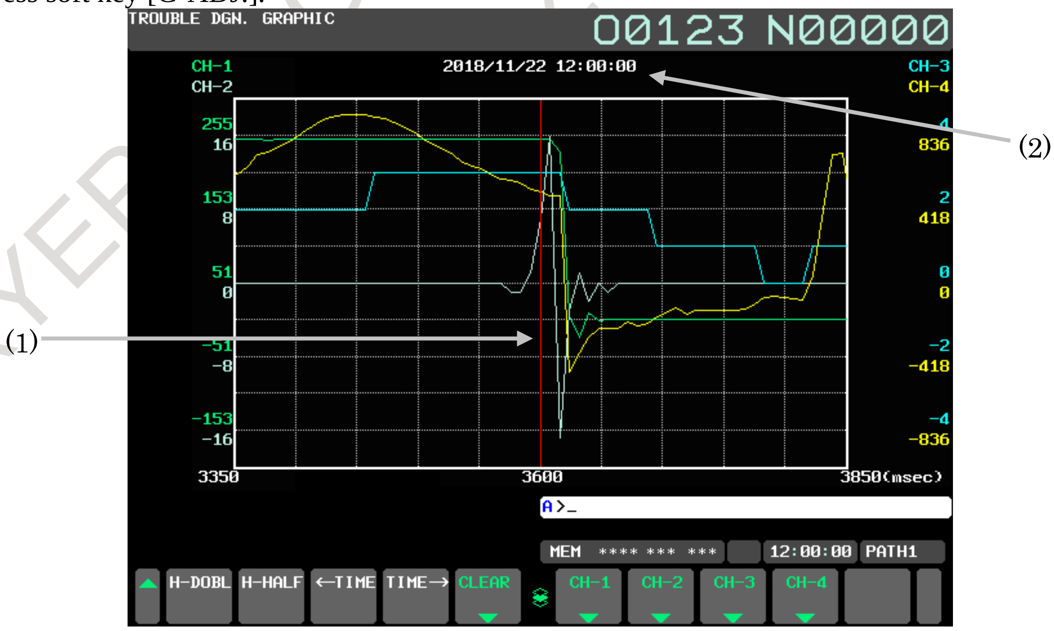


Fig.18.4.5 (a) Trouble diagnosis graphic screen (10.4-inch display unit)

- Contents of display

(1) Red vertical line shows the position (time) in which the alarm is occurred..
(2) The date when the alarm was occurred is displayed.

Display procedure of trouble diagnosis graphic screen (for 15/19-inch display unit)

Procedure

1 Press function key [?MESSAGE].
2 Press the vertical soft key [NEXT PAGE] until vertical soft key [W.GRPH] appears.
3 Press the vertical soft key [W.GRPH].
4 Press the vertical soft key [G-ADJ.].

Operation

- Change of position and magnification

When Soft key [G-ADJ.] is pressed, the following soft keys appear.

```
[<] [W.PRM ] [G-ADJ.] [TRB_LV] [      ] [(OPRT)] [  ]

[<] [H-DOBL] [H-HALF] [←TIME]  [TIME→] [      ] [+]

[<] [ CH-1 ] [ CH-2 ] [ CH-3 ] [ CH-4 ] [      ] [+]

[<] [WAV.EX] [WAV.RE] [WAV.↑]  [WAV.↓] [      ] [  ]
```

Soft key [H-DOBL]	: The length of the time base is expanded.
Soft key [H-HALF]	: The length of the time base is reduced.
Soft key [TIME←]	: The time base is moved to the left side.
Soft key [TIME→]	: The time base is moved to the right side.
Soft key [CH-1]	: The operation to the waveform of channel 1 is selected.
Soft key [CH-2]	: The operation to the waveform of channel 2 is selected.
Soft key [CH-3]	: The operation to the waveform of channel 3 is selected.
Soft key [CH-4]	: The operation to the waveform of channel 4 is selected.
Soft key [WAV.EX]	: The length of the spindle is expanded to the selected waveform.
Soft key [WAV.RE]	: The length of the spindle is reduced to the selected waveform.
Soft key [WAV.↑]	: The selected waveform is moved for above.
Soft key [WAV.↓]	: The selected waveform is moved below.

The time scale is a common scale for the channel 1,2,3 and 4.
The position and magnification of the height scale can be set for each channel.

Parameter setting for waveform display on the trouble diagnosis graphic screen

The data kind and the data unit of the waveform data displayed on the trouble diagnosis graphic screen are displayed and set on the trouble diagnosis parameter screen.

Display procedure of trouble diagnosis parameter screen (for 8.4/10.4-inch display unit)

Procedure

1. Press function key [MESSAGE].
2. Press the continuous menu key [▷] until soft key [W.GRPH] appears.
3. Press soft key [W.GRPH].
4. Press soft key [W.PRM].

Fig. 18.4.5 (b) Trouble diagnosis parameter screen (10.4-inch display unit)

Display procedure of trouble diagnosis parameter screen (for 15/19-inch display unit)

Procedure

1. Press function key [MESSAGE].
2. Press the vertical soft key [NEXT PAGE] until vertical soft key [W.GRPH] appears.
3. Press the vertical soft key [W.GRPH].
4. Press the vertical soft key [W.PRM].

Explanation

- Contents of display

CHANNEL :

Channel for waveform display. Maximum 4 channels can be used. Set data from the first channel sequentially.

AXIS :

Axis for waveform display. Manual setting is available. In some cases Axis data is set automatically for diagnosis in trouble diagnosis guidance screen.

DATA KIND :

Data kind for waveform display. Manual setting is available. In some cases data kind is set automatically for diagnosis in trouble diagnosis guidance screen.

DATA UNIT :

Unit of display data. This data is set automatically according to data kind. Manual setting is invalid.

FORECAST LV :

This data decides boundary value to output trouble forecast signal. Manual setting is available.

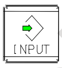
Guidance of setting each item :

The guidance to the above setting each item is displayed.

Operation

- Data Setting

1 Move cursor by Cursor key to the setting item.

2 Input number by MDI key and press input key 

to set data.

The content of the following each items can be input by the above operation.

AXIS :

In case of servo axis, input control axis number.
(Example) Set "1" for first servo axis.
In case of spindle axis, input "100 + spindle number".
(Example) Set "101" for first spindle axis.

> **NOTE**
> Set the index of an axis in the path system to the control axis number and the control spindle number for the multi path system.

DATA KIND :

Input data number value according to the following table.

Table 18.4.5 (a) Data of Servo motor

Data number	Data kind
1	Accumulated command pulse (pulse)
2	Accumulated feedback pulse (pulse)
3	Position error (pulse)
4	Reference counter (pulse)
5	Actual speed (1/min)
6	Command current (%)
7	Thermal simulation data (%)
8	Torque (%)

Data number	Data kind
9	Disturbance level (%)
10	Optional data
11	R-phase current (%)
12	Effective current (%)
13	Pulse coder AMR data
14	Optional data 2
20	SV U CURRENT (A)
21	SV V CURRENT (A)
22	PS L1-L2 VOLT. (V) (NOTE 3)
23	PS L2-L3 VOLT. (V) (NOTE 3)
24	PS L3-L1 VOLT. (V) (NOTE 3)
25	PS L1 CURRENT (A) (NOTE 3)
26	PS L2 CURRENT (A) (NOTE 3)
27	PS L3 CURRENT (A) (NOTE 3)
28	PS DC LINK VOLT. (V) (NOTE 3)

Table 18.4.5 (b) Data of Spindle motor

Data number	Data
15	Actual speed (1/min)
16	Load meter (%)
17	Position error (pulse)
18	Actual speed (1/min)/(80msec)
19	Load meter (%)/(80msec)
30	SP U CURRENT (A)
31	SP V CURRENT (A)
32	PS L1-L2 VOLT. (V) (NOTE 3)
33	PS L2-L3 VOLT. (V) (NOTE 3)
34	PS L3-L1 VOLT. (V) (NOTE 3)
35	PS L1 CURRENT (A) (NOTE 3)
36	PS L2 CURRENT (A) (NOTE 3)
37	PS L3 CURRENT (A) (NOTE 3)
38	PS DC LINK VOLT. (V) (NOTE 3)

NOTE

1 Each data kind is classified as follows as the combination to which the waveforms can be displayed at the same time.
- Data number :1 – 17
- Data number :18, 19
- Data number :20, 21, 30, 31
- Data number :22 - 28, 32 – 38

Therefore, if the values other than the same combination as the data kind of another channel are set when the data number is input to the data kind, Warning "DATA SETTING ERROR" is displayed.

2 The emergency must stop when the set point displays the data kind of waveforms of 20 or more number.

3 The displayed values as Voltage and Current are approximate value and they have some error. If you want to know the precise values, measure them by exclusive equipments.

18.4.6 Trouble forecast level setting screen

Trouble forecast level of the servo motor is set in this screen.
The breakdown level at the thermal simulation and disturbance level can be set.

Explanation

Thermal simulation

The calorific value of the motor and the amplifier is simulated based on a real current of the servo motor. The alarm of "SOFTTHERMAL(OVC)" is occurred so that the motor or the amplifier might cause over heat when this thermal simulation data reaches 100%.
Data value increases in the following cases.

1) Operating that frequency of acceleration and deceleration is high.
2) Operating to which state with very large cutting load continues.
3) The machine did not work according to the instruction though the motor output a full torque by the contact etc. of the machine and work.

Disturbance level

The disturbance level shows the torque that the motor receives from the machine excluding the torque to accelerate/decelerate the motor and the machine inertia in the output torque of the motor.

1) The machine came in contact with work.
2) The cutting load has increased.

An increase in these values is observed by this function, and warning can be occurred before the accident to the machine and occurrence of the alarm.

Display procedure of trouble forecast level setting screen (for 8.4/10.4-inch display unit)

Procedure

1 Press function key [MESSAGE].
2 Press the continuous menu key [▷] until soft key [W.GRPH] appears.
3 Press soft key [W.GRPH].
4 Press soft key [TRB LV].

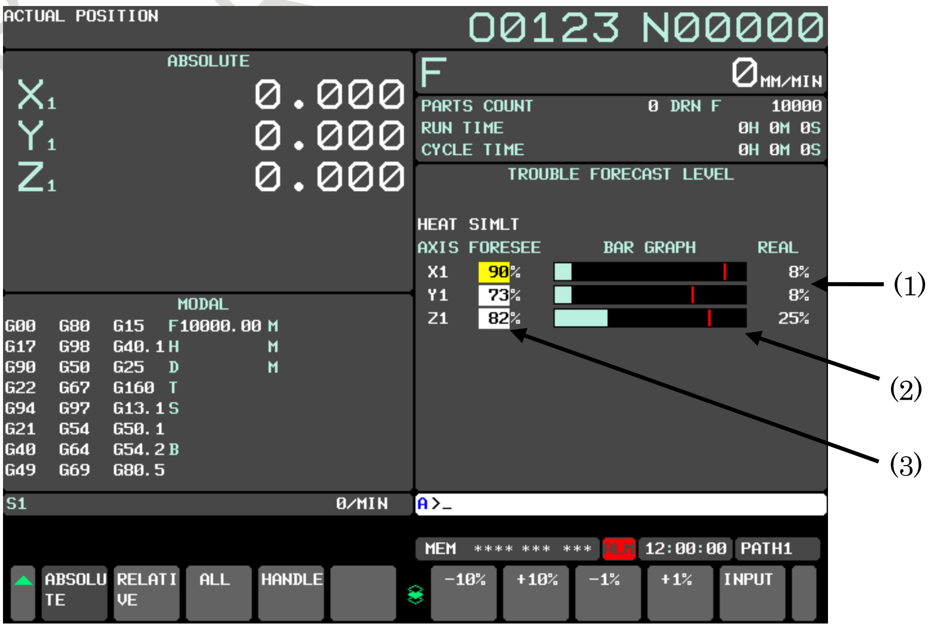


Fig.18.4.6 (a) Trouble forecast level setting screen (10.4-inch display unit)

- Contents of display

(1) Current value of thermal simulation/disturbance level is displayed with sign.

(2) Current value of thermal simulation or disturbance level is displayed by bar graphic.
Light Blue part : Current value of thermal simulation/disturbance level.
Red part : Trouble forecast level.

(3) Trouble forecast level.
The negative value is converted into the absolute value and a positive value is displayed.

Display procedure of trouble forecast level setting screen (for 15/19-inch display unit)

Procedure

1 Press function key .
2 Press the vertical soft key [NEXT PAGE] until vertical soft key [W.GRPH] appears.
3 Press the vertical soft key [W.GRPH].
4 Press the vertical soft key [TRB LV].

Operation

- Setting trouble forecast level

1 Select thermal simulation or disturbance level by page keys / .

2 Select axis by cursor keys / .

3 Input numerical value by MDI key and press input key or soft key [INPUT].

Trouble forecast level is input into parameter No.8860 and 8861.

- Change of trouble forecast level

When Soft key [TRB LV] is pressed, the following soft keys appear.

```
[<] [W.PRM ] [G-ADJ.] [TRB LV] [      ] [(OPRT)] [ ]
[<] [-10% ]  [+10% ]  [ -1% ]  [ +1% ]  [INPUT ] [ ]
```

Soft key [-10%]	: Subtract 10% from trouble forecast level.
Soft key [+10%]	: Add 10% to trouble forecast level.
Soft key [-1%]	: Subtract 1% from trouble forecast level.
Soft key [+1%]	: Add 1% to trouble forecast level.
Soft key [INPUT]	: Input trouble forecast level by MDI key.

- Trouble forecast signal

When thermal simulation or disturbance level of servo axis exceeds trouble forecast level, trouble forecast signal TDSML1 - 8 (Fn298)/ TDFTR1 - 8 (Fn299) is output.
Parameters TRSx (bit 0 of parameter No. 8855) / TRFx (bit 1 of parameter No. 8855) need to be set to 1 to perform the trouble forecast.

- Setting of trouble forecast level on the trouble diagnosis parameter screen

The trouble forecast level can be set even on the trouble diagnosis parameter screen.

1. The trouble diagnosis parameter screen is displayed.
2. The cursor is moved to the item set with the cursor key.
3. The axis is set to the servo axis.
4. The data kind is set at thermal simulation (7) or disturbance level (9).
5. Because the editing box next to "FORECAST LV" can be edited, the trouble forecast level that wants to be set to the axis is input.

Fig.18.4.6 (b) Setting of trouble forecast level on the trouble diagnosis parameter screen

18.4.7 Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8850								MDG

[Input type] Parameter input
[Data type] Bit

#0 MDG Trouble diagnosis function is:
0: Available.
1: Not available.

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

	#7	#6	#5	#4	#3	#2	#1	#0
8855							TRFx	TRSx

[Input type] Parameter input
[Data type] Bit axis

#0 TRSx In Trouble diagnosis function, trouble forecast of thermal simulation of servo axis is:
0: Not available.
1: Available. (Set the trouble forecast level to parameter No.8860.)

#1 TRFx In Trouble diagnosis function, trouble forecast of disturbance level of servo axis is:
0: Not available.
1: Available. (Set the trouble forecast level to parameter No.8861.)

8860	Trouble forecast level for thermal simulation

[Input type] Parameter input
[Data type] Word axis
[Unit of data] %
[Valid data range] 0 to 100
In Trouble diagnosis function, trouble forecast level for thermal simulation is set.

8861	Trouble forecast level for disturbance level

[Input type] Parameter input
[Data type] Word axis
[Unit of data] %
[Valid data range] 0 to 100
In Trouble diagnosis function, trouble forecast level for disturbance level is set.

18.4.8 Signal

Trouble forecast signals for thermal simulation TDSML1 - TDSML8 <Fn298>

[Classification] Output signal
[Function] The thermal simulation data is notified to exceed the trouble forecast level.
[Output condition] When the thermal simulation data of servo is a value that is bigger than the trouble forecast level, this signal is "1".

Trouble forecast signals for disturbance level TDFTR1 - TDFTR8 <Fn299>

[Classification] Output signal
[Function] The disturbance level is notified to exceed the trouble forecast level.
[Output condition] When the disturbance level of servo is a value that is bigger than the trouble forecast level, this signal is "1".

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn298	TDSML8	TDSML7	TDSML6	TDSML5	TDSML4	TDSML3	TDSML2	TDSML1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn299	TDFTR8	TDFTR7	TDFTR6	TDFTR5	TDFTR4	TDFTR3	TDFTR2	TDFTR1

18.4.9 Restrictions

- The use of the I/O Link β*i* amplifier

The alarm occurred by the motor with the I/O Link β*i* amplifier cannot be diagnosed.

- Using with waveform diagnosis display function together

When data of the machine signal or the servo/spindle are traced by waveform diagnosis display function, the waveform data displayed on the trouble diagnosis graphic screen is not sampled.

- This function on the NC Guide / NC Guide Pro

This function is a not available on the NC Guide / NC Guide Pro.

- The use of the servo guide

When the servo guide is used for the condition of either the following, spindle monitor data on the trouble diagnosis monitor screen and the spindle data of the trouble diagnosis graphic screen are not correctly displayed.

- When CNC systems of G321 or G331 are used.

18.5 MACHINE ALARM DIAGNOSIS

18.5.1 Outline

It is necessary to notify the operator by the alarm display and warning message in the machine tool when a peculiar problem to the machine such as the tool breakage and coolant shortage occurs.
The machine tool builder customizes by using the external alarm, the macro alarm, and the operator message function and a peculiar alarm and message to such a machine can be displayed.
In this function, the diagnostic information to specify the factor in addition can be added to the alarm and the message that the machine tool builder customized.
When a peculiar alarm to the machine etc. are occurred, the added diagnostic information is displayed on the guidance screen of the trouble diagnosis function and can be used to specify the cause.

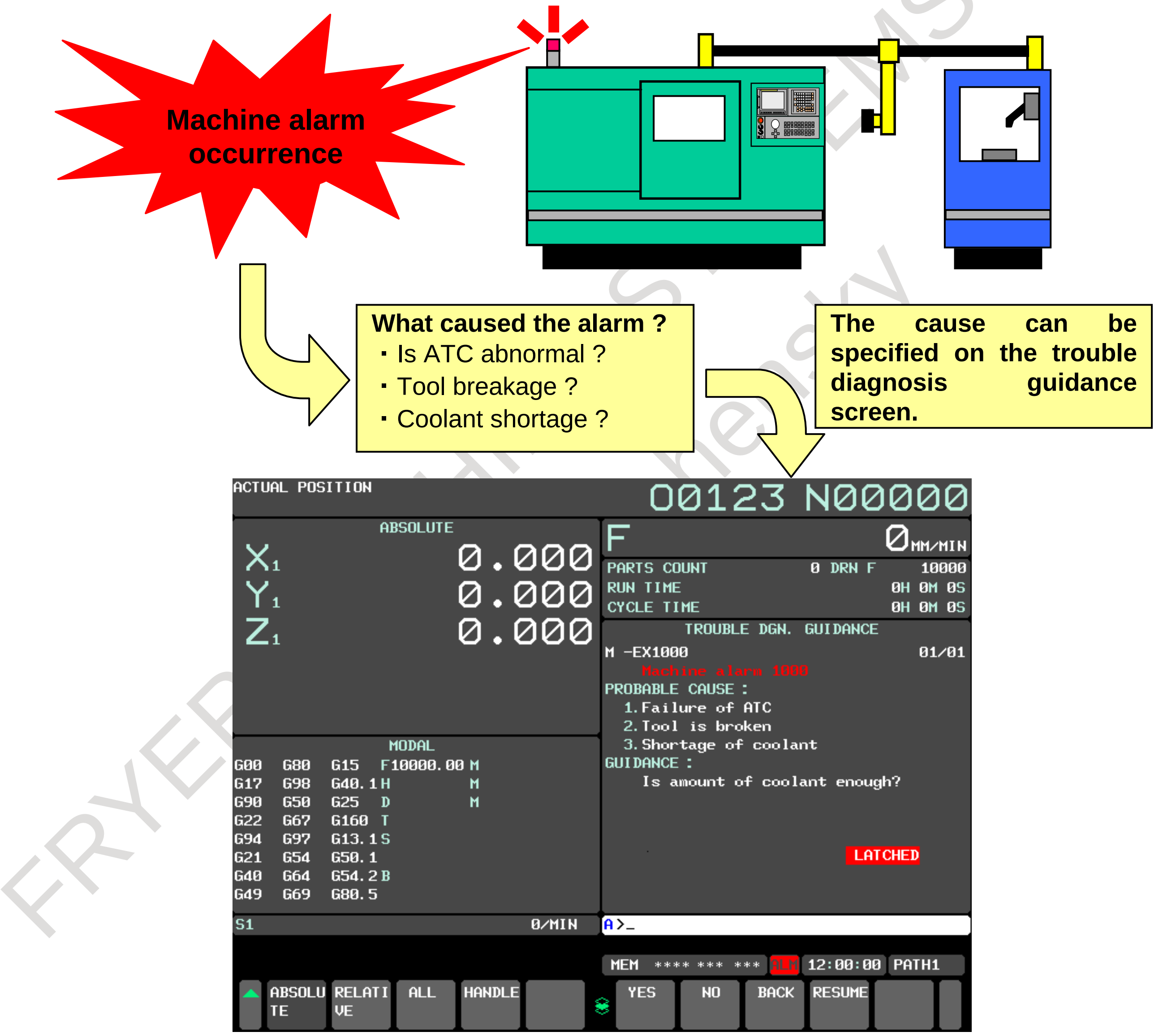


Fig.18.5.1 (a) Trouble diagnosis guidance screen (10.4-inch)

To add the diagnostic information on the screen, make the diagnostic message data to be displayed and convert it the memory card format file that can be read by CNC by using the PC tool "Guidance table for machine alarm diagnosis".
After the file is loaded into CNC, the additional diagnostic information can be displayed on the screen.

18.5.2 Additional alarm and operator message

The alarm and the operator message on the machine side that can be added to the trouble diagnosis message are as follows.

1. External alarm
2. Macro alarm
3. External operator message

18.5.3 Diagnosis Number

In the alarm and the operator message on the machine side, the alarm number and message number that can be diagnosed in the trouble diagnosis guidance screen are as follows.

1. External alarm

Table 18.5.3 (a) Available diagnosis alarm number (External alarm)

	Parameter EXA(No.6301#0)	
	=0	=1
Alarm number	1000 to 1999	0 to 4095

2. Macro alarm

Table 18.5.3 (b) Available diagnosis alarm number (Macro alarm)

	Parameter MCA(No.6008#1)	
	=0	=1
Alarm number	3000 to 3200	0 to 4095

3. External operator message

Table 18.5.3 (c) Available diagnosis message number (External operator message)

	Parameter EXM(No.6301#1)	
	=0	=1
Message number	2000 to 2999	0 to 4095

NOTE

1 Available diagnosis message number in the external operator message is a number according to the setting of parameter No.6310.

2 The above available diagnosis number is a range of the number displayed on the CNC screen, and it is different from the number actually set by the each function. Refer to the following manuals for the number actually set by the each function.
   - External alarm / External operator message
     Series 30*i*/31*i*/32*i*/35*i*-B PMC PROGRAMMING MANUAL (B-64513EN) ”INSTRUCTIONS RELATED TO CNC FUNCTIONS”
   - Macro alarm
     Series 0*i*-MODEL F Plus USER’S MANUAL (Common to Lathe System/Machining Center System) (B-64694EN) “CUSTOM MACRO“

18.5.4 Environment for Making Trouble Diagnosis Message

The following operating environment and the PC tool "Guidance table for machine alarm diagnosis" are necessary to make the trouble diagnosis message displayed on trouble diagnosis guidance screen.
Prepare the personal computer as follows, and install this tool according to the next chapter.

- Operating environment

Operating system	Microsoft® Windows® XP Professional or Microsoft® Windows Vista® Business
Memory	Windows® XP : More than 512M bytes Windows Vista® : More than 1G bytes
Free space in hard disk	More than 128M bytes
Necessary equipment	PC card adapter
Necessary application	Microsoft® Excel® 2007

* Microsoft, Windows, Windows Vista and Excel are the registered trademarks of Microsoft corporation in USA.

> **NOTE**
> "Guidance table for machine alarm diagnosis" is not included in the "machine alarm diagnosis function".
> Purchase "Guidance table for machine alarm diagnosis" CD (A08B-9010-J523#ZZ11) additionally.

18.5.5 Guidance Table for Machine Alarm Diagnosis

This tool is used to make the trouble diagnosis message displayed on the trouble diagnosis guidance screen.
Install this tool in the personal computer according to the following procedures, and make the trouble diagnosis message.

18.5.5.1 Install

The installation procedure of this tool is as follows:

1 Run setup.exe in the install CD (A08B-9010-J523#ZZ11).
2 In the following setup dialog box, set an appropriate installation folder and press the <Next> button.

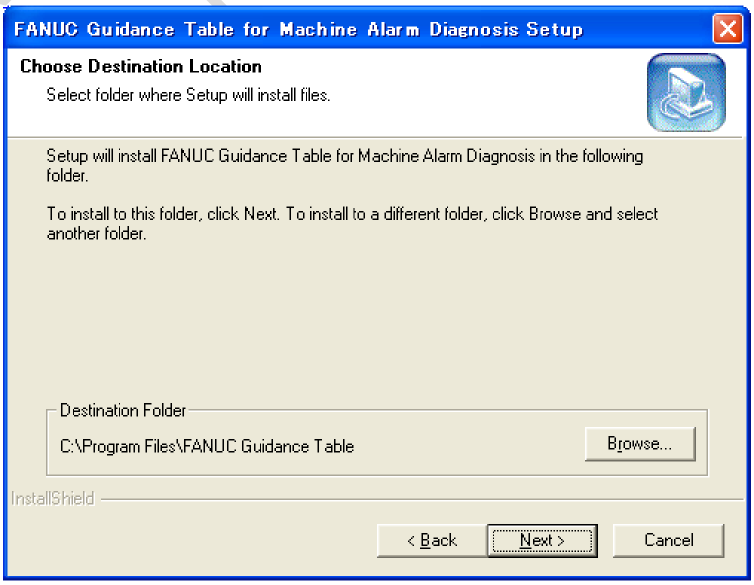


Fig. 18.5.5.1 (a)

NOTE
By pressing the <Browse> button, the installation folder can be changed.

3 By pressing the <Next> button in the dialog box for the confirmation of install, the installation will be started.
4 After the installation is finished normally, a message box to show the finish of the installation appears and "FANUC Guidance Table" is added in the Windows start menu.

18.5.5.2 Uninstall

The uninstall procedure of this tool is as follows:

1 Open the Windows control panel.
2 In the control panel, open the [Add or Remove Programs] and select "FANUC Guidance Table for Machine Alarm Diagnosis" from the currently installed programs.

NOTE
Open the [Programs and Features] in Windows Vista.

3 By pressing the <Remove (or Uninstall)> button, a message box for the confirmation of uninstall appears. By pressing the <Yes> button, this tool will be uninstalled.

18.5.5.3 Making a file to input trouble diagnosis messages

Making a Excel file in which trouble diagnosis messages are input is as follows:

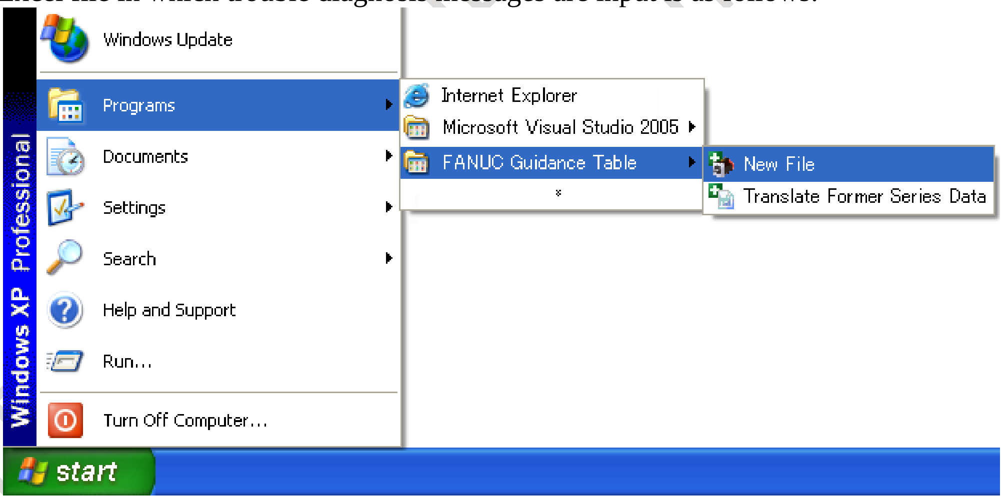


Fig. 18.5.5.3 (a)

1 Select [Program]→[FANUC Guidance Table]→[New File] in the Windows start menu, then the "New File" dialog box will be displayed.

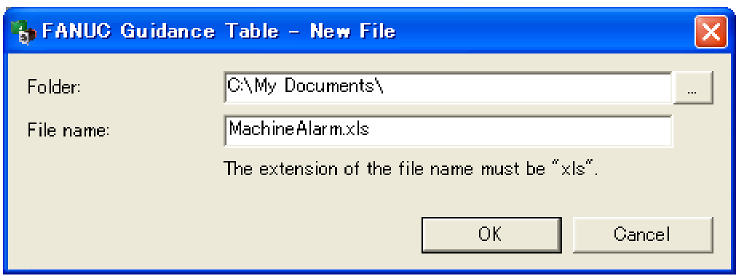


Fig. 18.5.5.3 (b)

2 In the following dialog box, input a destination folder and file name of the Excel file to be made.

NOTE

1 The extension of the file name must be "xls" (Excel book format). Any other extension is ignored and it is changed to "xls".

2 When making a new file is failed, the message box "A trouble diagnosis message file cannot be made." is displayed with one of the following messages about the cause of that failure.

- The specified file name is not correct.
- The specified file has already existed.
- (Destination folder) is not found.
- Files in (destination folder) cannot be written.
- The disk space in (destination folder) is not enough to make a new file.

3 Click the [OK] button, a Excel file will be made as the specified name in the specified folder.

The trouble diagnosis messages can be input by opening the new Excel file.

18.5.5.4 Structure of the file to input trouble diagnosis messages

Three are 2 sheets in the Excel file to input the trouble diagnosis messages.

- [Overall] sheet ・・・The sheet for overall operations and settings
- [Guidance] sheet ・・・The sheet for inputting message strings

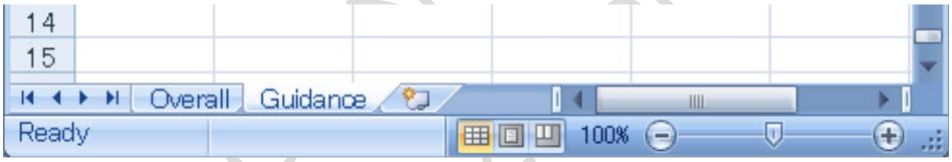


Fig.18.5.5.4 (a)

- Structure of [Overall] sheet

In the [Overall] sheet, you can make the trouble diagnosis messages by using the following operation buttons from (1) to (3).
And, you can switch the display language in sheets and change the settings for multi-languages.

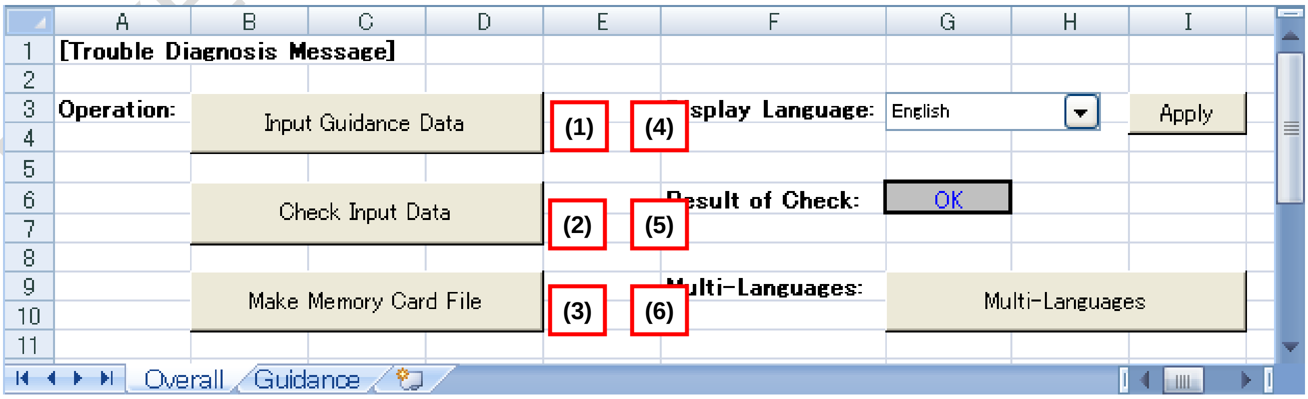


Fig.18.5.5.4 (b)

- Overview for each items

Item	Name	Description	Reference
(1)	Input Guidance Data	Displays the [Guidance] sheet to input the trouble diagnosis messages.	Chapter 18.5.6.1
(2)	Check Input Data	Checks all input data on the [Guidance] sheet as to checking for the range of number, the invalid letter and so on.	Chapter 18.5.6.2
(3)	Make Memory Card File	Makes a CNC readable memory card format file from input data in the [Guidance] sheet.	Chapter 18.5.6.3
(4)	Display Language	Switches the display language for sheets by selecting a language and pressing the [Apply] button. Either Japanese or English can be selected.	—
(5)	Result of Check	Displays the result of check for all input data.	Chapter 18.5.6.2
(6)	Multi-Languages	Changes the settings to input messages for multi-languages.	Chapter 18.5.7

- Structure of [Guidance] sheet

In the guidance sheet, you can input numbers and messages for each alarm or operator message.

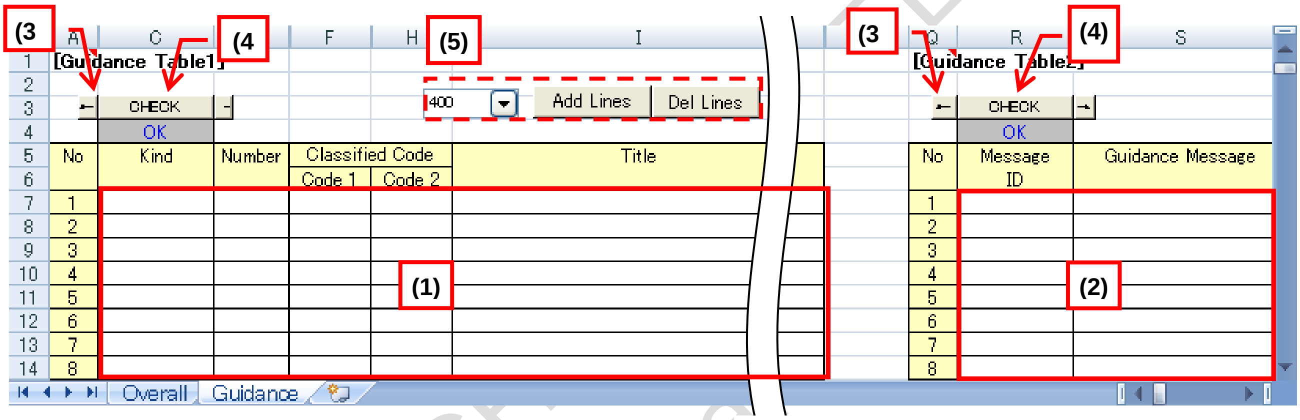


Fig.18.5.5.4 (c)

- Overview for each items

Item	Name	Description	Reference
(1)	Guidance Table 1	The table to input numbers and titles for alarm or operator message.	Chapter 18.5.6.1
(2)	Guidance Table 2	The table to input questions and instructions to an operator for alarm or operator message.	Chapter 18.5.6.1
(3)	Jump to Page	Switches the display between the guidance table 1 and 2.	—
(4)	Check Input Data	Checks input data for each guidance table.	Chapter 18.5.6.2
(5)	Set Line Number	Adds or deletes lines by a number specified in the combo box at the bottom of the guidance table 1 or 2. Note that all line number in the guidance table is from the minimum 30 to the maximum 2000.	—

18.5.6 Making Trouble Diagnosis Messages

Make the trouble diagnosis messages by using the following operation buttons in order from (1) to (3) on the [Overall] sheet.
The brief of making procedure is as follows:

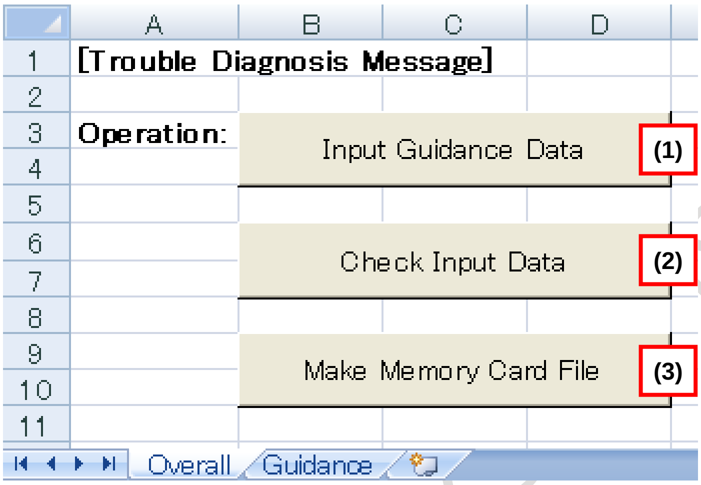


Fig. 18.5.6 (a)

(1) Input Guidance Data
Displays the [Guidance] sheet to input trouble diagnosis messages.
(2) Check Input Data
Checks all input data on the [Guidance] sheet

(3) Make Memory Card File
Makes a memory card format file from input data in the [Guidance] sheet.

The details for each operation are as follows:

18.5.6.1 Input Guidance Data

By clicking the [Input Guidance Data] button on the [Overall] sheet, the [Guidance] sheet will be displayed.
Input data on the guidance table 1 and 2 according to the descriptions of the next section.
The data that is input on each guidance table is displayed on the trouble diagnosis guidance screen as the following figure.

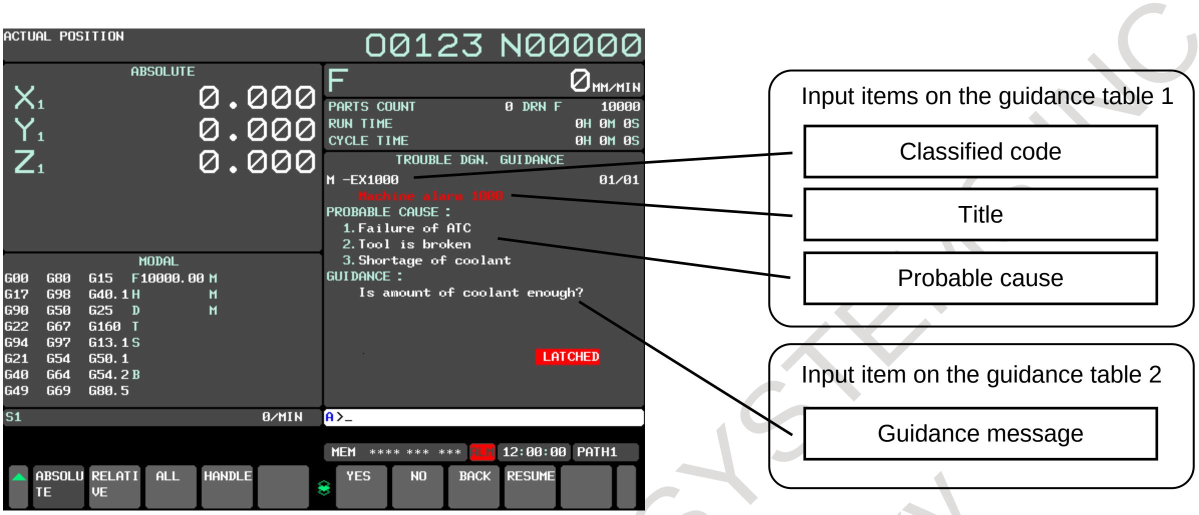


Fig. 18.5.6.1 (a)

Table 18.5.6.1 (a) Display item of trouble diagnosis guidance screen

Display Item	Description
Classified code	Classified code and number for the current alarm or operator message
Title	Title for the current alarm or operator message
Probable cause	Cause of the current alarm or operator message
Guidance message	Confirmation to determine cause of an alarm, or an instruction about an operator message

> **NOTE**
> The character that can be displayed on the CNC screen can be used for each display item. As for details, refer the following manuals.
> Series 0*i*-MODEL F Plus USER’S MANUAL (Common to Lathe System/Machining Center System) (B-64694EN)
> “APPENDIX F. CHARACTER-TO-CODES CORRESPONDENCE TABLE “

- About input data in the guidance table 1

No	Kind	Number	Classified Code		Title	Probable Cause			Message ID	Additional Information (Reserved)
			Code 1	Code 2		First Line	Second Line	Third Line		
1										
2	(1)	(2)	(3)		(4)		(5)		(6)	(7)
3										

Table 18.5.6.1 (b) Description of items in the guidance table 1

Item	Description	Content of item	Num. of letters
(1)	Kind	Input a kind for an external alarm, a macro alarm or an operator message. Specify one of the following strings. EX : External alarm MC : Macro alarm OP : External operator message **NOTE** If a kind other than above is input, checking input data is failed.	2 English capital letters
(2)	Number	Input a number for an external alarm, a macro alarm or an operator message.	Within 4 digits
(3)	Classified Code	Input 2 types of code, "Code 1" and "Code 2". "Code 1 – Code 2" is displayed at the first line on the trouble diagnosis guidance screen.	
		Code 1: Input a classified code for an alarm or an operator message.	Within 2 letters (1 in multi-bytes letters)
		Code 2: Input a number for an alarm or an operator messages.	Within 16 letters (8 in multi-bytes letters)
		NOTE 1 It is not necessary to make each classified code to be the same as (1)kind and (2)number. 2 A character string "NC" is reserved as the code 1 of CNC alarm. Do not use "NC" as the code 1.	
(4)	Title	Input a string displayed at the second line on the trouble diagnosis guidance screen.	Within 32 letters (16 in multi-bytes letters)
(5)	Probable Cause	Input a string displayed at the "Probable cause" part on the trouble diagnosis guidance screen.	Within 32 letters (16 in multi-bytes letters) per line (maximum 3 lines)
(6)	Message ID	Input an ID of a message displayed on the "Guidance Message" part of the trouble diagnosis guidance screen at first when an alarm occurs. Ex. The first guidance message for the external alarm message alarm No.1000 : EX1000A **NOTE** As for the message ID, see the description of the next description " About input data in the guidance table 2".	Within 8 letters (English capital and numerical letters)
(7)	Additional information (Reserved)	Do not input in the item.	

- About input data in the guidance table 2

No	Message ID	Guidance Message	Next Message ID		Notes
			YES	NO	
1					
2	(1)	(2)	(3)		(4)
3					

Table 18.5.6.1 (c) Description of items in the guidance table 2

Item	Description	Content of item	Num. of letters
(1)	Message ID	It is a string to identify to one of the guidance messages in (2). Ex.1. The first message ID for the external alarm No.1000 : EX1000A Ex.2. The second message ID for the external alarm No.1000 : EX1000B **NOTE** The character string that starts by "M"+"Number" is not used as message ID. Because this shows the destination of the jump of the machine side to guidance table 2 from the guidance table for the CNC alarm. Therefore, do not use the character string that starts by "M"+"Number".	Within 8 letters (English capital and numerical letters)
(2)	Guidance Message	It is a message displayed on the "Guidance Message" part of the trouble diagnosis guidance screen. Input a question, an instruction and so on to an operator. **NOTE** When the guidance message is shown on the trouble diagnosis screen, a new line code is inserted to the message per 32 letters (16 in multi-bytes letters) automatically. If you want to insert a new line code at a free position, input "¥n" that is not included as the number of letters. Moreover, the line number of the message is within 4 including the above automatic new line.	Within 128 letters (64 in multi-bytes letters)
(3)	Next Message ID	Input a message ID that is jumped to when pressing the soft key [Yes] or [No] for each question of guidance message. Input "-1" when there is no message ID to be jumped.	Within 8 letters (English capital and numerical letters)
(4)	Notes	Free notes can be written.	

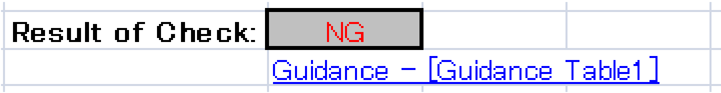
18.5.6.2 Checking input data

By clinking the [Check Input Data] button on the [Overall] sheet, all input data on the guidance sheet will be checked as to the range of number, the number of letters, the invalid letter and the line number of strings.

After the check is completed, the string of the [Result of Check] on the [Overall] sheet is changed from "Unchecked" to "OK".

> **NOTE**
> After "OK" is displayed, it might return to "Unchecked" by editing on the [Guidance] sheet again.

If the check is failed, the string of the [Result of Check] is changed to "NG" and a name of sheet and guidance table where an error occurs is shown in a cell under the [Result of Check].



The guidance table where an error occurs is displayed by clicking the name of the guidance table.

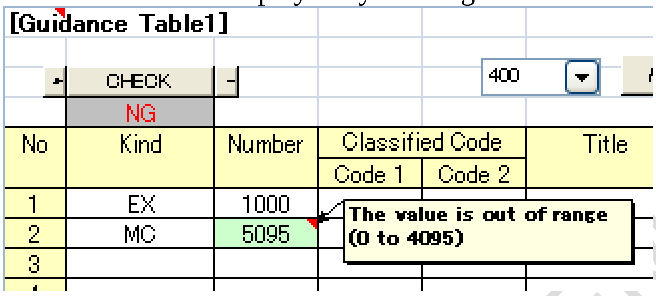


Fig. 18.5.6.2 (a)

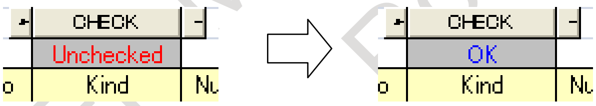
In the guidance table, the color of an error cell is changed to green and an error message is attached to the error cell as a commnet of the cell like the above figure. The list of the error messages is as follows:

- Input a number.
- The value is out of range (n to m).
- Input a string within n characters (m in multi-bytes characters).
- The line number is over limit n.
- The message referenced by the ID is not found.
- The kind(s) cannot be specified.

※m and n are numbers, s is a string

NOTE

1. The input data can be checked for an individual guidance table by clicking the [CHECK] button displayed at the top of the guidance table.
   In this case, the result of the check is displayed under the [CHECK] button like the following figure.



2. In case that errors occurs on two or more guidance tables, the [Result of Check] on the [Overall] sheet shows the name of the guidance table where the first error occurs. As for the result of check for each guidance table, see the display under the [CHECK] button.

Moreover, the color of the cell changes into light blue when the character string that shows the destination of the jump of the machine side to guidance table 2 from the guidance table for the CNC alarm is input to message ID.

Example)

No	Message ID
1	EX1000_A
2	EX1001_A
3	M205
4	

18.5.6.3 Making a memory card format file

By clicking the [Make Memory Card File] button on the [Overall] sheet, the following dialog box will be displayed.

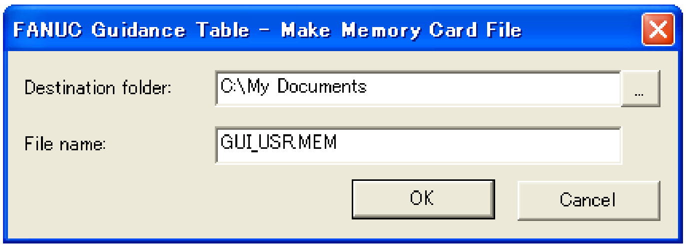


Fig. 18.5.6.3 (a)

Make a CNC readable memory card format file as follows:

1 Input a folder name where you want to make a file in the [Destination folder].

2 Input a file name that you want to make in the [File name].
" GUI_USR.MEM " is input by default.
When you input a file name without its extension, the extension "MEM" is added to the file name automatically.

3 Click the [OK] button.

After making the file is completed, the message box "Making the memory card format file is completed." will be displayed.

> **NOTE**
> 1 The available file name is a short file name with alphabetic and numerical letters (8.3 MS-DOS format).
> Other name format cannot be read by CNC.
> 2 If a file having the same name as the specified one has already existed in the destination folder, the file always will be overwritten.
> 3 When making the memory card format file is failed, the message box "The memory card format file cannot be made." is displayed with one of the following messages about the cause of that failure.
> - The input data have not been checked on (sheet name) – guidance table (1 or 2)
> - There are error(s) of the input data on (sheet name) – guidance table (1 or 2)
> - The specified file name is not correct.
> - (Destination folder) is not found.
> - Files in (destination folder) cannot be written.
> - The disk space in (destination folder) is not enough to make a new file.

By loading the file to CNC with the CNC BOOT function, the trouble diagnosis message can be displayed on the trouble diagnosis guidance screen when the relevant external alarm, macro alarm or operator message happens.

> **NOTE**
> As for the CNC BOOT function, see the following manuals.
> Series 30*i*/31*i*/32*i*-B MAINTENANCE MANUAL (B-64485EN) "APPENDIX C. BOOT SYSTEM"

18.5.6.4 Jump from CNC guidance table to MTB's guidance table

As a result of the diagnosis of the CNC alarm, the diagnosis of the machine side might be needed.
There is special message ID that jumps from the guidance table of the CNC alarm to the guidance table on the machine side for the diagnosis of such a case.
It is possible to jump to the diagnosis of MTB's guidance table after the diagnosis of the CNC alarm if the guidance table is made by using Message ID shown by "M"+"Number" in the following tables.

Table 18.5.6.4 (a) Reserved message ID

No.	Message ID	Title	Probable cause
1	M205	RIGID MODE DI SIGNAL OFF	The rigid mode DI signal (G061.0) is not set to 1 when the rigid tapping is executed.
2	M409	DETECT ABNORMAL TORQUE	A mechanical collision or twist occurred, resulting in a load torque higher than a normal operation value.
3	M410	EXCESS ERROR (STOP)	A mechanical collision or twist occurred, disabling an axis from reaching a target position.
4	M411	EXCESS ERROR (MOVING)	A mechanical collision or twist occurred, disabling an axis from moving.
5	M420	SYNC TORQUE EXCESS	Two axes to be moved synchronously lost mechanical synchronism with each other, resulting in a large torque difference.
6	M421	EXCESS ERROR (SEMI-FULL)	With a closed-loop machine, a shift occurred between the motion of the motor and the motion of the separate detector by a cause such as a mechanical twist.
7	M436	SOFT THERMAL (OVC)	A mechanical collision or twist occurred, resulting in a large load and the flow of an excessive current.

NOTE
Message ID shown by "M"+"Number" other than existing in the above table is not available.

18.5.7 Making Messages for Multi-languages

To change the language of messages displayed in the trouble diagnosis guidance screen complying with the display language of CNC, it is necessary to input messages for each language and make the memory card format file.
The messages for each display language are input on dedicated sheets for multi-languages.

NOTE
The messages in the [Guidance] sheet are assumed as English.
If you need messages other than English, make messages as follows:

18.5.7.1 Making sheets for multi-languages

Making sheets for multi-languages is as follows:

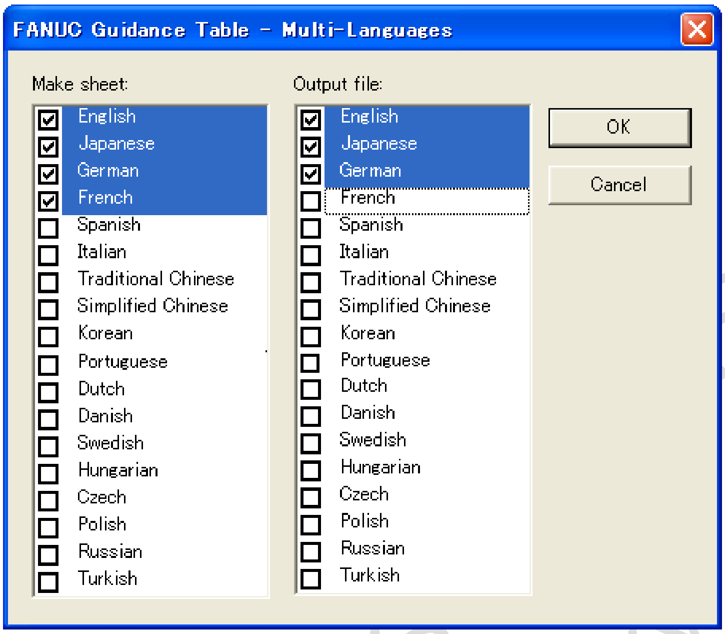


Fig. 18.5.7.1 (a)

1 Click the [Multi-Languages] button on the [Overall] sheet. The following dialog box will be displayed.

2 Turn on a check box for a language that you want to make in the [Make sheet] list. Two or more check boxes can be turned on.
After turning on the check box in the [Make sheet] list, the check box for the corresponding language can be operated in the [Output file] list.

3 In the [Output file] list, turn on a check box for a language that you want to output to your CNC readable memory card format file at the same time. Up to 5 check boxes other than English can be turned on. 6 or more check boxes cannot be turned on.

4 Click the [OK] button.
The sheets that are selected in the item 2 will be made like the following figure.

Sheets for multi-languages

NOTE
1 The supported languages are 18 ones shown in the previous dialog box.
2 The number of languages that can be output to the file of the memory card format at the same time is six languages (English + five national languages).
3 Once a sheet for multi-languages is made, the language of the [Guidance] sheet assumed as English. And, in the previous dialog box, the check box for English in the [Make sheet] and [Output file] list is turned on by default and they cannot be operated.
4 If you input other than English messages in the [Guidance] sheet before making the sheet for multi-languages, move the messages in the [Guidance] sheet to a sheet for an appropriate language by cut and paste operation of Excel.
5 To remove the sheet for multi-languages, turn off the check box for the language that you want to remove in the [Make sheet] list in the previous dialog box.

18.5.7.2 Inputting data in the sheet for multi-languages

The way to input data on the sheet for multi-languages is the same as one on the [Guidance] sheet that has already explained in the section "Making trouble diagnosis messages". However, data can be input for the 3 items only – Title, Probable cause and Guidance message.
As for data other than the 3 times, data in the [Guidance] sheet is copied to here and displayed.

No	Kind	Number	Classified Code		Title	Probable Cause			Message ID
			Code 1	Code 2		First Line	Second Line	Third Line	
1	EX	1000							
2	MC	3000							
3									

Fig. 18.5.7.2 (a) Sheet for multi-languages : Guidance table 1

No	Message ID	Guidance Message	Next Message ID	
			YES	NO
1				
2				
3				

Fig. 18.5.7.2 (b) Sheet for multi-languages : Guidance table 2

After inputting data in the sheet for multi-languages, check the input data and make the memory card format file according to the procedure described in "Making trouble diagnosis messages".

NOTE
The display language of the trouble diagnosis guidance screen is switched automatically according to the CNC display language.
However, when there is no message for the CNC display language in the memory card file, the input messages in the [Guidance] sheet are displayed.

18.5.8 Notice

- Excel's file of trouble diagnosis message

When you use "Guidance table for machine alarm diagnosis", change the setting of the macro function to "Enable" in the Trust Center of Excel 2007. If the setting is "Disable", the trouble diagnosis message cannot be made.
To enable the macro function by setting a path in the "Trusted Locations" of the Trust Center, set the folder where the file to input trouble diagnosis messages should be located and also the install folder of the "Guidance table for machine alarm diagnosis".

18.5.9 Translating Data Used with the Former Series (Series 0*i* /0*i* Mate-B/C, Series 16*i* /18*i* /21*i*-B)

The Excel file and the memory card file used with the former series (Series 0*i* /0*i* Mate-B/C, Series 16*i* /18*i* /21*i*-B) cannot be used with the current target series (Series 30*i* /31*i*/32*i*-B, 31*i*-B5) of this function.
Translate the Excel file used with the former series and make the memory card format file again according to the following description.

NOTE
As for the way to make the trouble diagnosis message for Series 0*i* /0*i* Mate-B/C, Series 16*i* /18*i* /21*i*-B, see the manual "Trouble Diagnosis Specifications" (A-78500) and "Machine Alarm Diagnosis Specifications" (A-78622)

1 Select [Program]→[FANUC Guidance Table]→[Translate Former Series Data] in the Windows start menu.

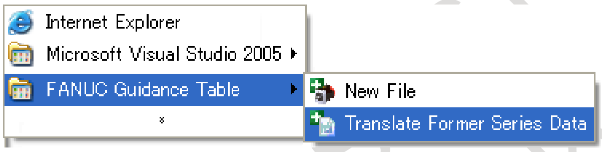


Fig. 18.5.9 (a)

2 In the displayed dialog box, input a Excel file name used with the former series in the [Source file name] and input a translated file name in the [Destination file name].

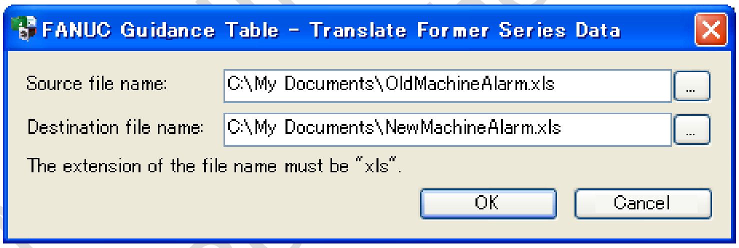


Fig. 18.5.9 (b)

3 Click the [OK] button, then the translated Excel file that is specified in the [Destination file name] will be made.

4 With the translated Excel file, check input data and make the memory card format file according to the section "Making trouble diagnosis messages".

NOTE

1. Input a full path in the [Source file name] and [Destination file name]. The path name cannot be omitted. The extension of the file name must be "xls" (Excel book format). Any other extension is ignored and it is changed to "xls".
2. The "Kind" in the guidance table 1 is input automatically complying with the "Number" during the translation.
   No. 1000 to 1999 → EX (External alarm)
   No. 2000 to 2999 → OP (External operator message)
   No. 3000 to 3200 → MC (Macro alarm)
3. When translating data is failed, the message box "The data cannot be translated." is displayed with one of the following messages about the cause of that failure.
   - (Source file name) is not found.
   - (Destination folder name) is not found.
   - Files in (destination folder) cannot be written.
   - The disk space in (destination folder) is not enough to make a new file
   - (Destination file name) has already existed.
4. If you use separated Excel files for Japanese and English with the former series, translate both files and make a sheet to input Japanese messages in the English file. Then, move the data form the Japanese file to the sheet to input Japanese in the English file by cut and paste operation of Excel so as to combine two files to one.

19 MAINTENANCE FUNCTION

19.1 MACHINE STATE MONITORING FUNCTION

19.1.1 Overview

This function monitors state of machine.
When trouble of the machine occurs, CNC information such as the operation history, the position and the feedrate at that time is saved in the CNC memory.
The saved CNC information can be used to investigate the cause of machine trouble.

Unexpected disturbance torque detection function, servo/spindle alarm, and a ladder program made by machine tool builders monitor the state of machine.
In the ladder program, the shock value is monitored by the Multi-Sensor Unit (MSU). When trouble of the machine such as the spindle collision occurs, the ladder program notifies CNC that the trouble has occurred.
When CNC is notified the trouble occurrence, CNC saves the information on that time in the CNC memory.
Machine tool builders or maintenance members investigate the cause of generation of trouble by using the saved CNC information.
The maximum save number of CNC information is 100. So, the past trouble can be investigated.

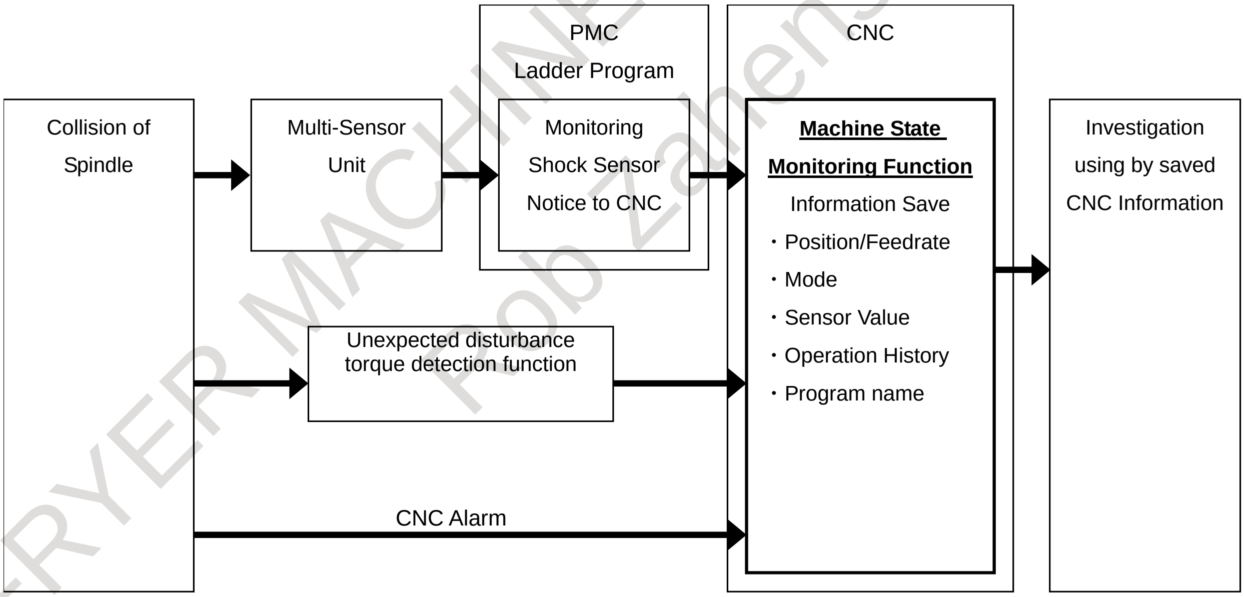


Fig. 19.1.1 (a) Summary of machine state monitoring function (Ex. collision of spindle)

Machine state monitoring function has the following two screens.
(1) Machine state monitoring screen
(2) Machine state history screen

In the machine state monitoring screen, four bytes PMC signals that set by parameter and the signal of the Multi-Sensor Unit can be monitored.

In the machine state history screen, saved CNC state and signal value can be displayed when match status history save condition occurs (machine status history save request signal set to 1, unexpected abnormal torque or servo/spindle alarm are detected). So, the signal of Multi-Sensor Unit and the CNC state such as position and program at the time when a spindle collides can be checked.
In addition, displayed information can be output to external devices.
When CNC saved information exceeded 100, CNC information is overwritten from the old one. CNC Information is remained for eight days or more, so number of saving information each day is restricted.

NOTE

Please refer to the following manuals for Multi-Sensor Unit and unexpected disturbance torque detection function.

- Multi-Sensor Unit:
  FANUC Series 0*i*-MODEL F Plus (HARDWARE) (B-64693EN)
- Unexpected Disturbance Torque Detection Function:
  FANUC Series 0*i*-MODEL F (FUNCTION) (B-64693EN-1)
  “2 PREPARATIONS FOR OPERATION 2.9UNEXPECTED DISTURBANCE TORQUE DETECTION FUNCTION”

19.1.2 Monitoring by CNC Software

CNC software monitors following three statuses and when at least one trouble is detected, CNC saves the information.

1. unexpected disturbance torque
2. servo alarm
3. spindle alarm

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11352	ABT	SVT	SPT					

[Input type] Parameter input
[Data type] Bit path

#5 SPT When spindle alarm occurred, recording CNC information by machine state monitoring is :
0: Disabled.
1: Enabled.

#6 SVT When servo alarm occurred, recording CNC information by machine state monitoring is :
0: Disabled.
1: Enabled.

#7 ABT When unexpected abnormal torque is detected, recording CNC information by machine state monitoring is :
0: Disabled.
1: Enabled.

19.1.3 Monitoring by Ladder Program

A PMC ladder program made by machine tool builders monitors the state of machine.
Machine tool builders deicide abnormal criteria according to the purpose such as spindle collision and machine overheat or the machine characteristic.

Example of abnormal detection)
(1) Detection of spindle collision
- The shock value that read by the Multi-Sensor Unit exceeds the assumption value.
- The load current value that read by the unexpected disturbance torque detection function exceeds the assumption value.

(2) Detection of machine overheat
- The temperature value that read by the Multi-Sensor Unit exceeds the assumption value.

When the trouble is detected, the ladder program sets machine state history save request signal MSUREQ <G0534.6> to “1” to notify CNC the occurrence of trouble.

19.1.4 Saving the Machine State

When the machine trouble is detected by machine state history save request signal MSUREQ <G0534.6>, servo alarm, spindle alarm or unexpected disturbance torque detection function, CNC saves the following information in the CNC memory.

The information can be saved up to 100.
If the new information is saved when history number is max, the oldest data is deleted and latest information is saved.

The saved CNC information cannot be deleted by the screen operation.

Table 19.1.4 (a) Saved CNC Information List

Item	Remarks
Time	Year, Month, Day, Hour, Minute, Second
PMC Signal	4bytes
Machine Position	All Axis, All Path
Absolute Position	All Axis, All Path
Feedrate	All Path
Spindle Speed	All Axis, All Path
Program Name (O Number)	Executing program name, All Path
Block Number	Block number of executing program, All Path
Mode	All Path
Modal	M, S, T, B (Without G code) All Path
Operation History	Data of 1/20 of normal operation history. About 400 MDI keys. The number of MDI keys decreases when a history with large size such as the alarm history is saved.
Non-Save Number	Number of information not saved. (refer to 22.1.4.1)
Multi-Sensor Unit Signals	Maximum 4
Save trigger	SIGNAL : signal MSUREQ ABTRQ : unexpected abnormal torque is detected SV ALM : servo alarm SP ALM : spindle alarm

The machine state at the time when trouble occurs can be checked by the above saved information.

Example)

(1) Spindle collision during automatic operation
If the operation mode is automatic operation such as MEM when the trouble occurs, the operator states can be clarified by checking the saved following data.
- The program name and its block number.
- The machine position and the absolute position.
- The Modal.

(2) Spindle collision during manual operation
If the operation mode is manual operation such as JOG mode when trouble is occurs, the operator states can be clarified by checking the saved following data.
- The machine position and the absolute position.
- The operation history that the CNC parameter is changed.

19.1.4.1 Restriction of saving

To save CNC information for eight or more days, so number of saving information each day is restricted.

Potential Saving Number

Potential Saving Number means the number of limits that can be preserved in a day.
Potential Saving Number is given basically 10 times a day, but remained Potential Saving Number is carried over next day. (Maximum 30)
CNC information is remained eight or more days in CNC memory. (Refer to Example.1 to 3)
Potential Saving Number is displayed on diagnosis data No. 4500.

If number of histories that can be saved per day are exceeded, the history at that time will not be saved even if the machine state history saving condition is satisfied.
The number of history records that were not saved will be recorded in the history that will be saved next time as the number of unsaved histories.

Non-Save Number

Non-Save Number is the number that CNC information was not saved because of restriction of saving even if save request condition is fulfilled(signal MSUREQ<G0534.6> changing into “1”, detecting unexpected abnormal torque, spindle alarm or servo alarm)
Non-Save Number is included in CNC information that will be saved next time.
The maximum value of Non-Save Number is 65535.
Non-Save Number is displayed on Machine state history screen. (Refer to the below figure.)
If “*” displayed at the right of detect time, it means the value of Non-Save Number is 1 or more.
If Non-Save Number is 0, “*” is not displayed.
In details, refer to 22.1.8 Machine state history screen.

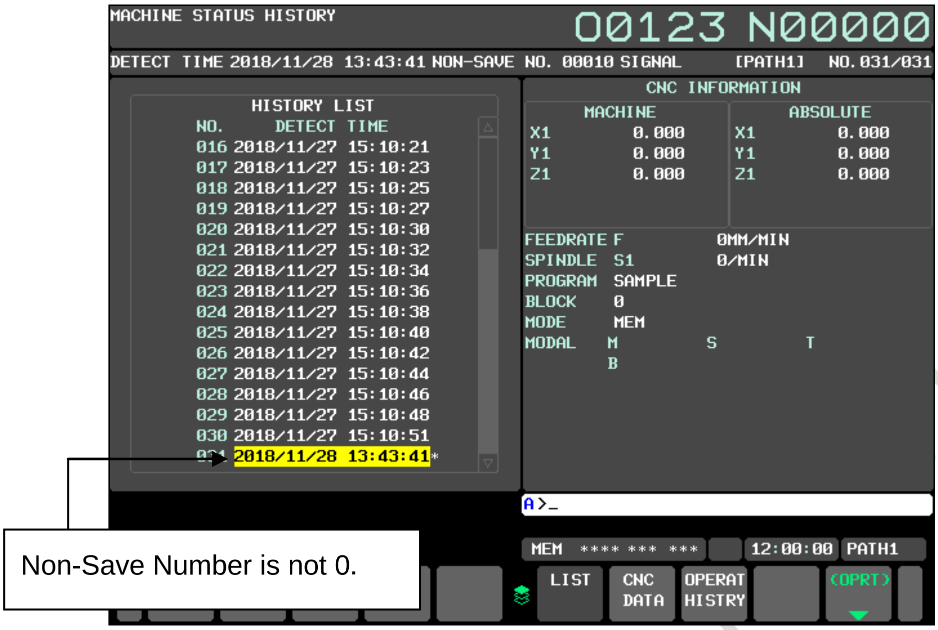


Fig. 19.1.4.1 (a) Non-Save Number

Example.1 (Non-Save Number)

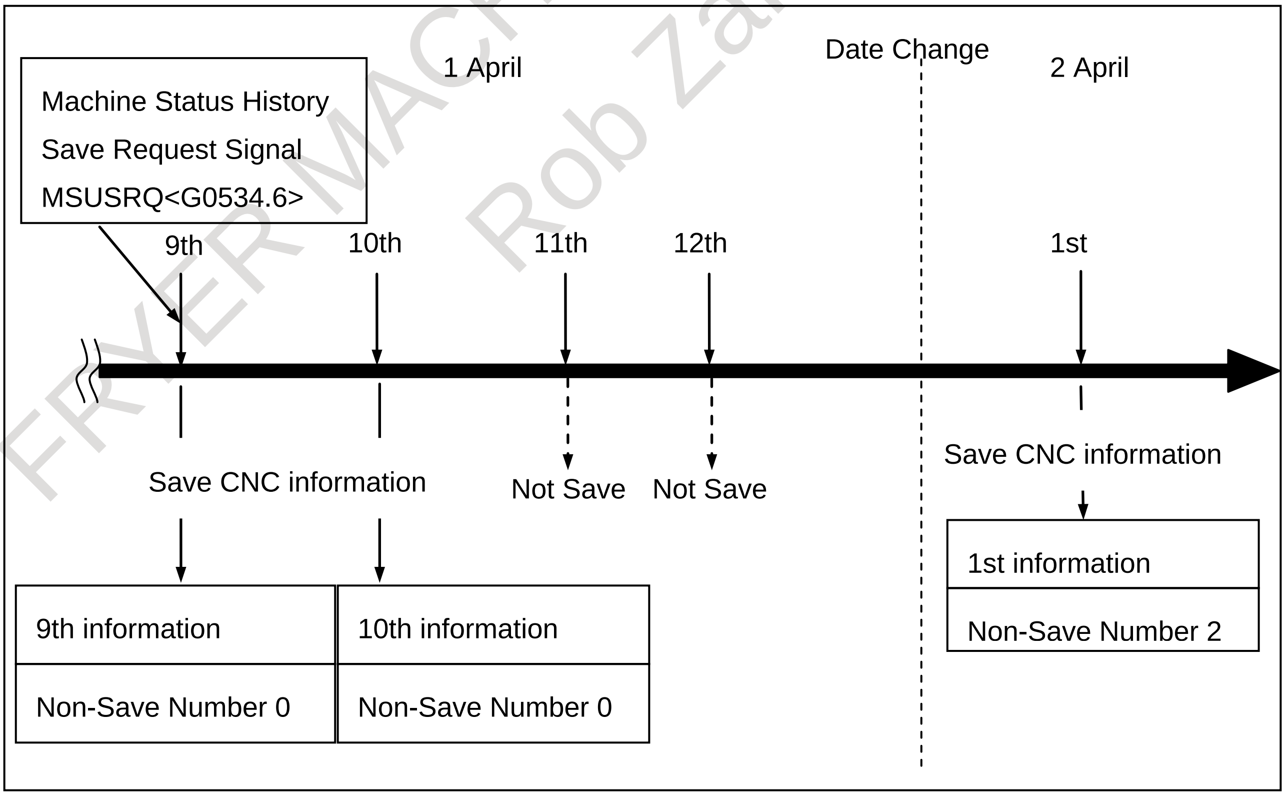
This is the case MSUREQ changes into “1” twelve times on 1 April, and change into “1” only one time 2 April.

On 1 April, Potential Saving Number was 10, and MSUREQ had changed into “1” twelve times.

From 1st to 10th CNC information was saved, but 11th and 12th had not been saved, because of restriction.

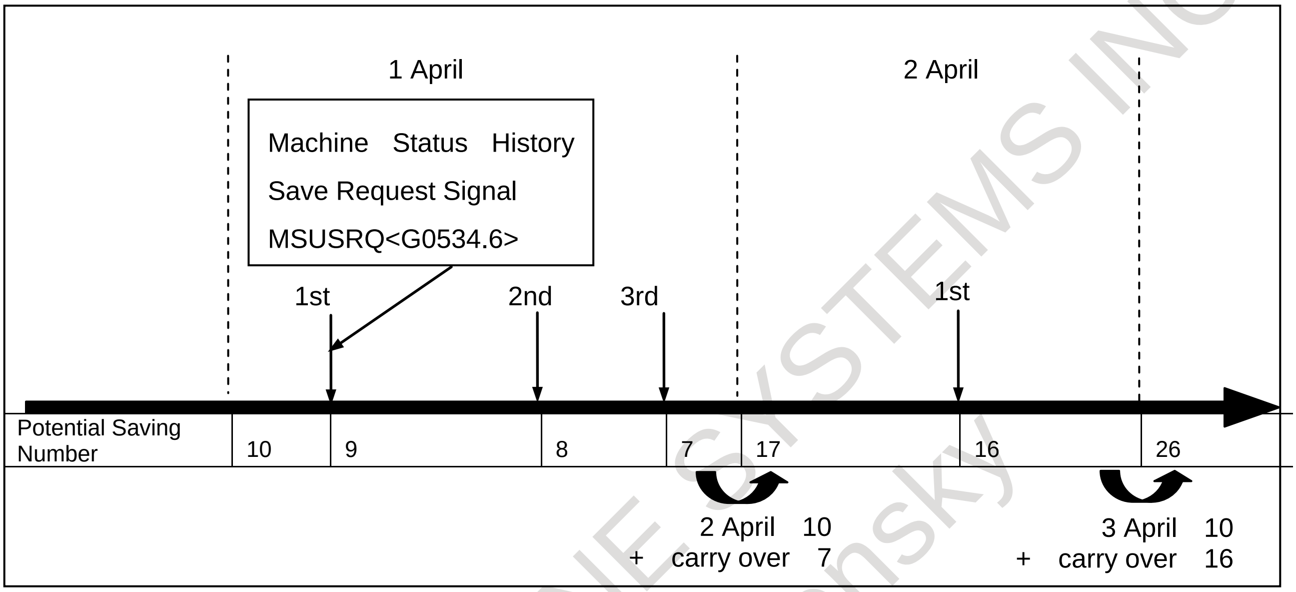
On 2 April, MSUREQ had changed “1” only one time.

In this case, the value of Non-Save Number is saved is 2(11th and 12th on 1 April).



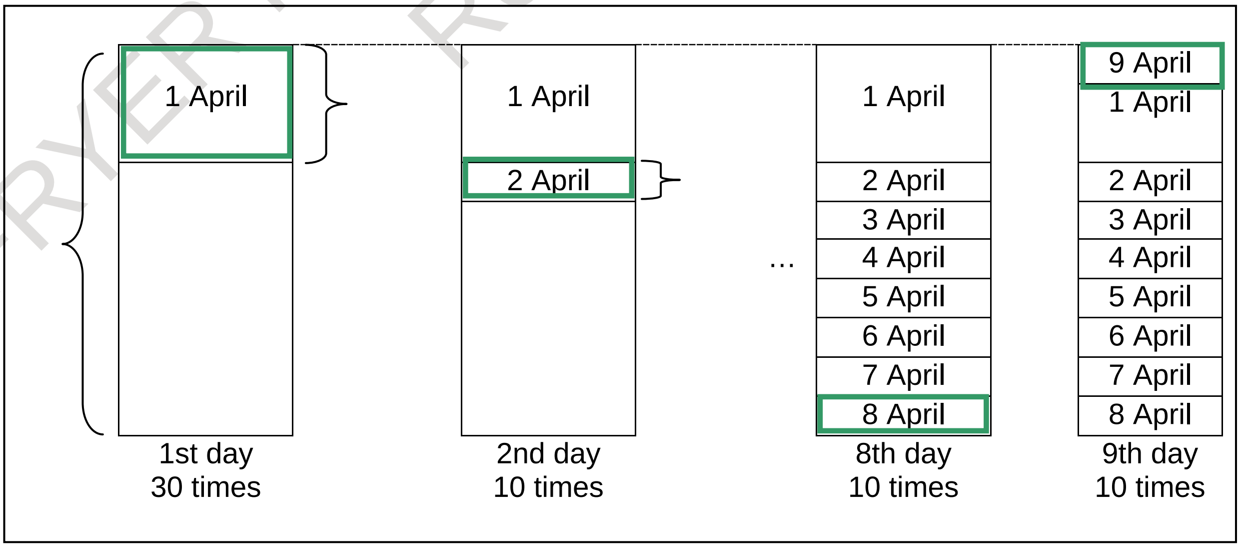
Example.2 (Carry over of Potential Saving Number)

This is the case of carry over of Potential Saving Number.
On 1 April, Potential Saving Number was 10 and CNC information was saved 3 times.
Potential Saving Number was carried over next day.
On 2 April, Potential Saving Number was 17 as a result of carry over from 1 April, and CNC information was saved only 1 time.
On 3 April, Potential Saving Number was 26 as a result of carry over from 2 April.
Max of carry over of Potential Saving Number is 30.



Example.3 (Saving 100 information by the shortest days)

This is the example of saving 100 information in the shortest days.
CNC information was saved 30 times on 1 April.
Next 7 days, information was saved 10 times a day.
On 8 April, there were CNC information from April 1, and information of 1 April still remained.
Like this, information remains for at least 8 days.
9 April the old information on 1 April is overwritten.



19.1.5 Setting

19.1.5.1 PMC signal

Four bytes data can be set as PMC signal.
The address is set by the parameter.

The PMC path number of signal No.1 to 4 is set to parameter No.11378 to No.11381.
The value and the PMC path number that can be set are as follows.

No.11378 to 11381	PMC Path Number
0	Unsetting
1	1st PMC
2	2nd PMC
3	3rd PMC
4	4th PMC
5	5th PMC
9	Dual Check Safety

The address kind of signal No.1 to 4 is set to parameter No.11382 to No.11385.
The value and the address kind that can be set are as follows.

No.11382 to 11385	Address Kind
0	Unsetting
1	X
2	Y
3	G
4	F
5	A
6	R
7	T
8	K
9	C
10	D

The address number of signal No.1 to 4 is set to parameter No.11386 to No.11389.
The range that can be set is decided depending on the address kind.
Refer to PMC Programming Manual (B-64513EN) for detail.

Example:

Signal No	Address	PMC Path Number	Address Kind	Address Number
1	1:X10	No.11378 = 1	No.11382 = 1(X)	No.11386 = 10
2	1:Y20	No.11379 = 1	No.11383 = 2(Y)	No.11387 = 20
3	2:R30	No.11380 = 2	No.11384 = 6(R)	No.11388 = 30
4	3:D40	No.11381 = 3	No.11385 = 10(D)	No.11389 = 40

19.1.5.2 Multi-Sensor Unit

The Multi-Sensor Unit is connected with CNC by I/O Link.
When the Multi-Sensor Unit is detected in the power on, CNC automatically sets the address of the sensor signal.
Therefore, the setting by MTB is unnecessary.

NOTE
The allocation setting of I/O Link is necessary.

19.1.5.3 Delay time for operation history saving

The alarm history can be saved in the operation history.
So, the operation history that contains the generated alarm can be saved by this function.
However, the gap might be caused in timing between the machine status history save request condition is fulfilled and the alarm is saved into operation history.
In this case, even if CNC saves the operation history when the machine status history save request condition is fulfilled, the operation history doesn't contain the alarm generated in the immediately after that.
To prevent this problem, the delay time for operation history save is set.
The delay time for the operation history save is set by parameter No.24302.

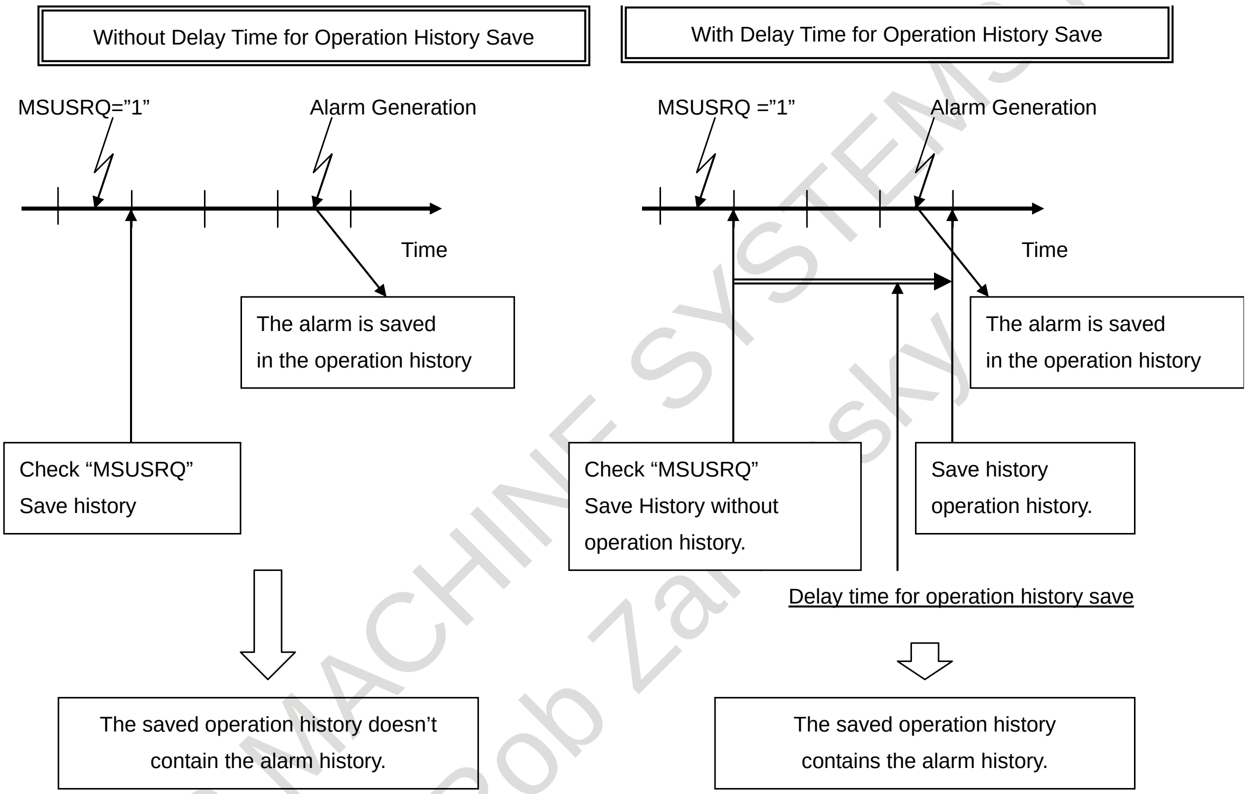


Fig. 19.1.5.3 (a) Delay time for operation history saving

If the parameter No.24302 is –1 or less, the delay time is 0.
If the parameter No.24302 is 0, the delay time is 160ms.
If the parameter No.24302 is 1 or more, the delay time is (No.24302 × 16)ms.
For example, if the parameter No.24302 is 5, the delay time is 80ms.

19.1.5.4 Machine state history saving response signal (MSUSAS)

Machine State History Saving Response Signal MSUSAS <F545.6> is response signal of MSUSRQ<G534.6>.If the machine status history start saving by other reason such as detecting spindle alarm, MSUSAS doesn't become "1".

When CNC detects MSUSRQ<G534.6> changing into "1", CNC changes MSUSAS<F545.6> into "1".
Even if starting saving machine state history when condition MSUSQR<G0534.6> is not set to 1, MSUSAS<F0545.6> is not turned to 1.
Afterwards, behavior of CNC is different depend on the value of Potential Saving Number.

− When Potential Saving Number is one or more
  (1) When CNC detects MSUSRQ changing into “1”, CNC begin to save information and change MSUSAS into “1”.
  (2) Ladder program confirm that MSUSAS changed into “1”, please change MSUSRQ into “0”.
  (3) After information saving is ended, if MSUSRQ is “0”, MSUSAS is changing into “0”.

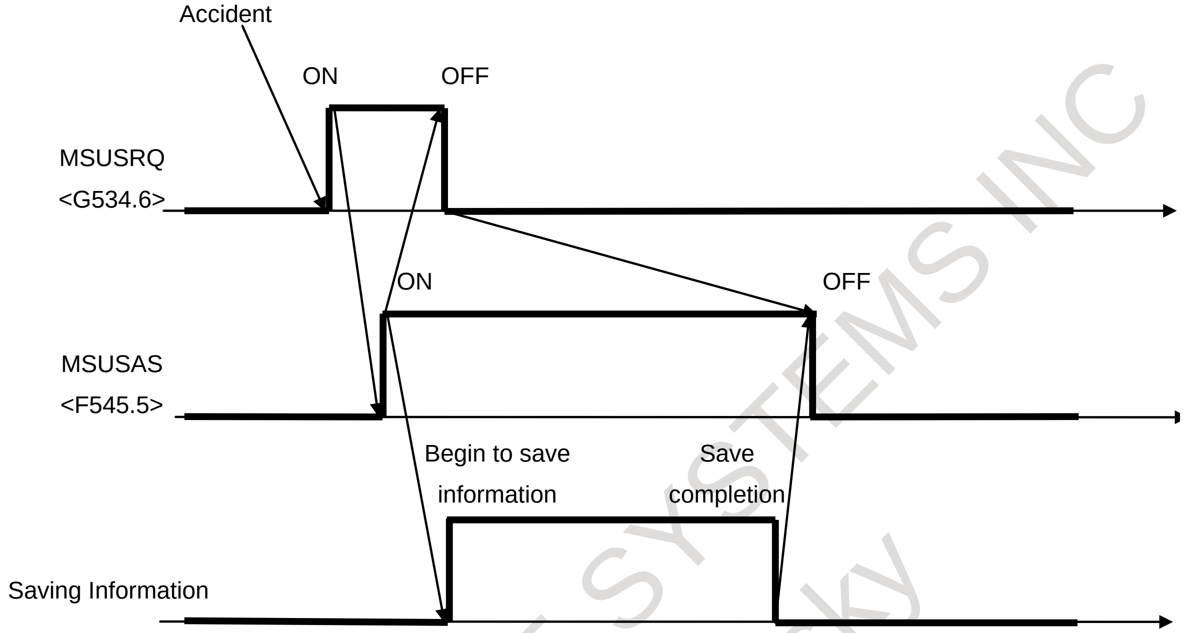


Fig. 19.1.5.4 (a) Machine State History Save Request Signal and Machine State History Saving Response Signal

− When Potential Saving Number is zero
  (1) When CNC detects MSUSRQ changing into “1”, CNC change MSUSAS into “1”, but CNC is not saving information.
  (2) Ladder program confirm that MSUSAS is changed into “1”, please change MSUSRQ into “0”.
  (3) If MSUSRQ is 0, MSUSAS changing into “0”.

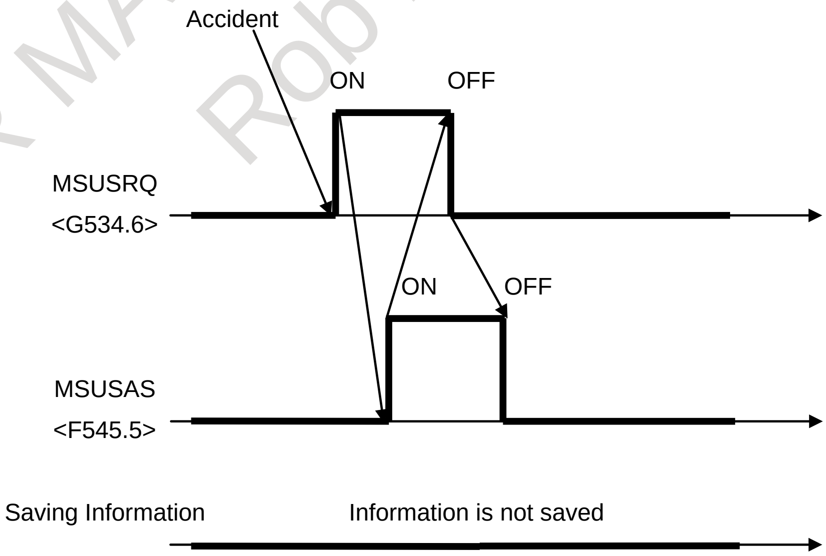


Fig. 19.1.5.4 (b) Machine State History Save Request Signal and Machine State History Saving Response Signal

Signal

Machine State History Save Request Signal MSUSRQ<G0534.6>

[Classification] Input signal
[Function] The save of the machine state history is requested.
[Operation] When this signal becomes 1, the machine state history is saved.

Machine State History Saving Response Signal MSUSAS<F0545.5>

[Classification] Output signal
[Function] Response to the machine status history save request signal (MSUSRQ)
[Output cond.] This signal becomes 1 when:
- Detect MSUSRQ is “1”.

It becomes 0 when:
Behavior is different depend on the information was saved or not.
- The machine state history is saving: Detect MSUSRQ is “0” and saving is ended.
- The machine state history is not saving: Detect MSUSRQ is “0”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0534		MSUSRQ						
F0545			MSUSAS					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11372	MSH				MSM			

[Input type] Parameter input
[Data type] Bit

#3 MSM The machine state monitoring screen
0: is not displayed.
1: is displayed.

#7 MSH The machine state history screen
0: is not displayed.
1: is displayed.

11378	PMC path number of PMC signal No.1 for the machine state monitoring function
to	
11381	PMC path number of PMC signal No.4 for the machine state monitoring function

[Input type] Parameter input
[Data type] Byte
[Data range] 0 to 5 and 9

The PMC path number of the PMC signal for the machine state monitoring function is set.
The value and the PMC path number that can be set are as follows.

No.11378 to 11381	PMC Path Number
0	Unsetting
1	1st PMC
2	2nd PMC
3	3rd PMC
4	4th PMC

No.11378 to 11381	PMC Path Number
5	5th PMC
9	Dual Check Safety

11382	Address kind of PMC signal No.1 for the machine state monitoring function
to	
11385	Address kind of PMC signal No.4 for the machine state monitoring function

[Input type] Parameter input
[Data type] Byte
[Data range] 0 to 10
Address kind of PMC signal for the machine state monitoring function is set.
The value and the address kind that can be set are as follows.

No.11382 to 11385	Address Kind
0	Unsetting
1	X
2	Y
3	G
4	F
5	A
6	R
7	T
8	K
9	C
10	D

11386	Address number of PMC signal No.1 for the machine state monitoring function
to	
11389	Address number of PMC signal No.4 for the machine state monitoring function

[Input type] Parameter input
[Data type] 2 Word
[Data range] Refer to PMC Programming Manual (B-64513EN) for detail.
Address number of PMC signal for the machine state monitoring function is set.

24302	Delay time from trigger signal to storing operation history for the machine state monitoring function

[Input type] Parameter input
[Data type] Word
[Data range] -1 to 32767
This parameter sets delay time from turning on trigger signal to beginning preservation of the operation history in the machine state monitoring function.

Parameter No.24302	Delay Time
-1	0msec
0	160msec
1 to 32767	Parameter No.24302×16msec

Diagnosis Data

4500	Potential Saving Number

[Valid data range] 0 to 30
Potential Saving Number is displayed.

19.1.6 Display the CNC Information

In the CNC screen, the PMC signal and the signal of the Multi-Sensor Unit can be monitored and the CNC information at the time when the trouble of machine occurs is checked.
The screen for the confirmation has the following two kinds.

- Machine state monitoring screen: Display of current PMC signals and Multi-Sensor Unit information.
- Machine state history screen: Display of CNC information at the time when the trouble of machine occurs.

19.1.7 Machine state monitoring screen

In the machine state monitoring screen, four PMC signals that set by parameter and the signal of the Multi-Sensor Unit can be monitored.

The machine state monitoring screen is displayed by the following procedures.

1. Press function key SYSTEM.
2. Press the continuous menu key several times to display the soft key [MACHIN MONITR]
3. Press the soft key [MACHIN MONITR] to display the machine state monitoring screen.

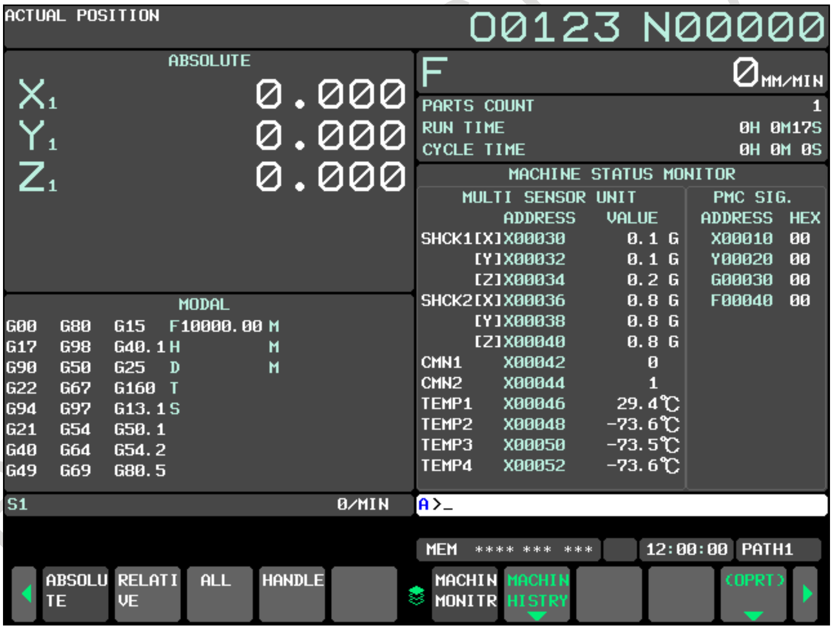


Fig. 19.1.7 (a) Machine state monitoring screen(With Multi-Sensor Unit)

> **NOTE**
> This screen is displayed only when bit 3 (MSM) of parameter No.11372 is 1.

19.1.7.1 Monitoring of PMC signals

In the machine state monitoring screen, four PMC signals that set by parameter can be monitored.

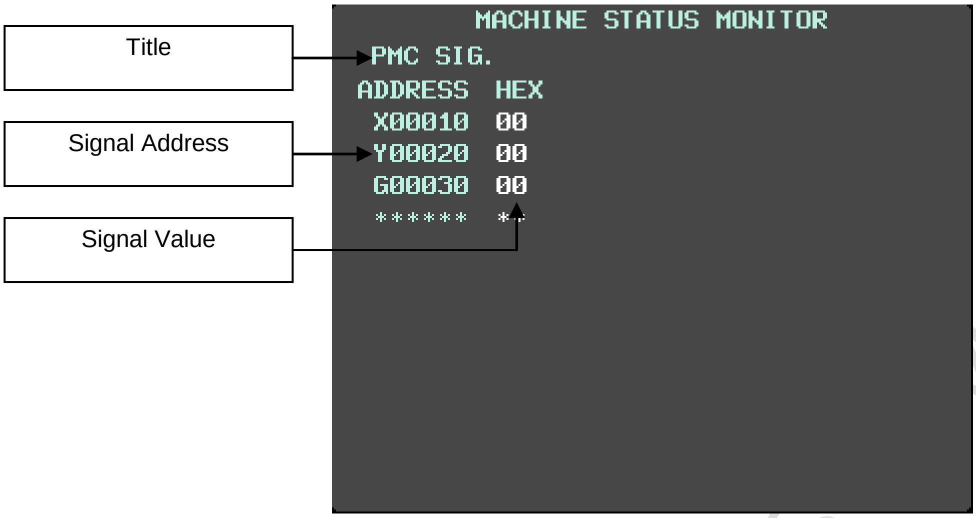


Fig. 19.1.7.1 (a) Monitoring of PMC signals

Details of the content of the display are as follows.

PMC Signal Title

The title of PMC signal that can be set by the parameter is displayed.
The PMC signal can be set only by four bytes or less.

PMC Signal Address

The address of PMC signal set by the parameter is displayed.
If the parameter is not set, “****” is displayed.
In the case of multi PMC system, the PMC path number is displayed ahead of the address. (Refer to Fig. 19.1.7.1 (b))

PMC Signal Value

The real-time value of PMC signal set by the parameter is displayed by Hex.
If the parameter is not set, “****” is displayed.

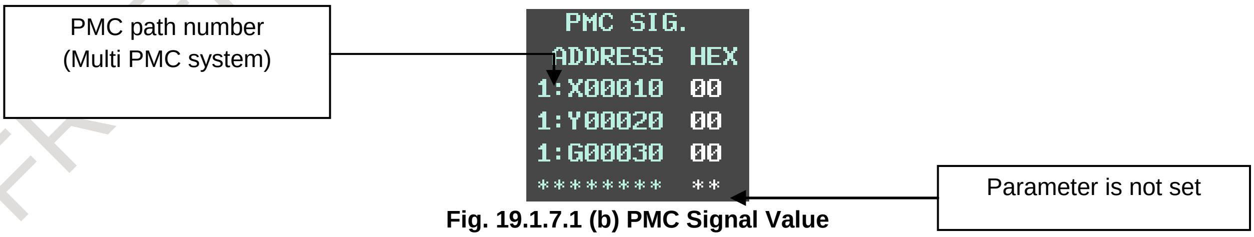


Fig. 19.1.7.1 (b) PMC Signal Value

19.1.7.2 With Multi-Sensor Unit

When Multi-Sensor Unit is connected, machine state monitoring screen is displayed as follows.

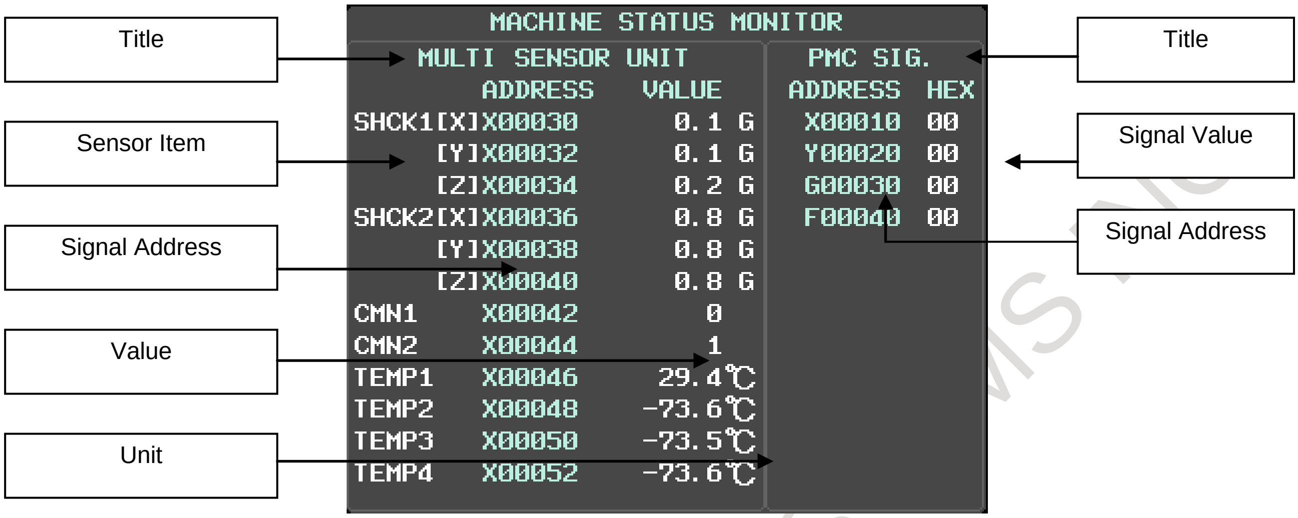


Fig. 19.1.7.2 (a) With Multi-Sensor Unit

Details of the content of the display are as follows.

Multi-Sensor Unit Title

The title of Multi-Sensor Unit title is displayed.

For the multi PMC system

In the case of multi PMC system, the PMC path number of the Multi-Sensor Unit is displayed next to the title.

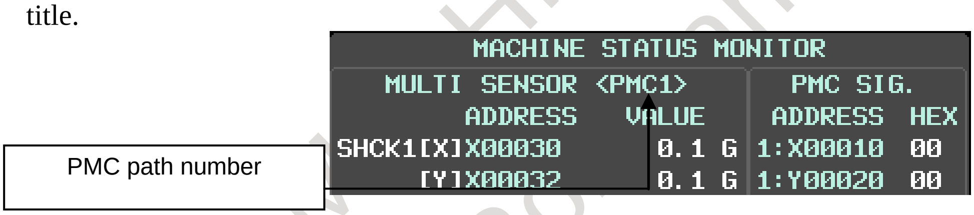


Fig. 19.1.7.2 (b) Multi PMC System

For the plural Multi-Sensor Units

The Multi-Sensor Unit can be connected up to four.
In the case of the plural Multi-Sensor Units, the displayed Multi-Sensor Unit number (MSU No.) and all connected number of MSU are displayed.
The Multi-Sensor Unit number is counted from the one near CNC to four among units connected with I/O Link.

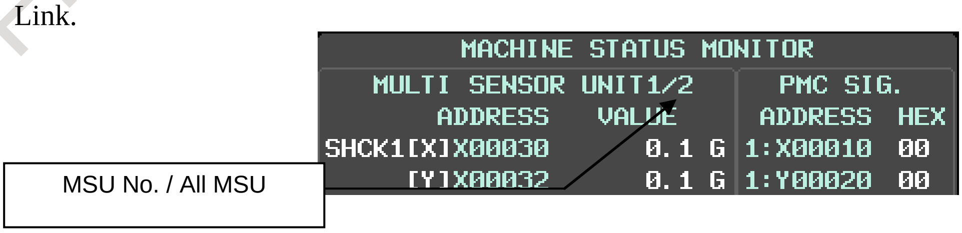


Fig. 19.1.7.2 (c) plural Multi-Sensor Units

For multi PMC system and the plural Multi-Sensor Units

In the case of multi PMC system and the plural Multi-Sensor Units, the displayed MSU No, all connected number of MSU and the PMC path number of the Multi-Sensor Units are displayed next to the title.

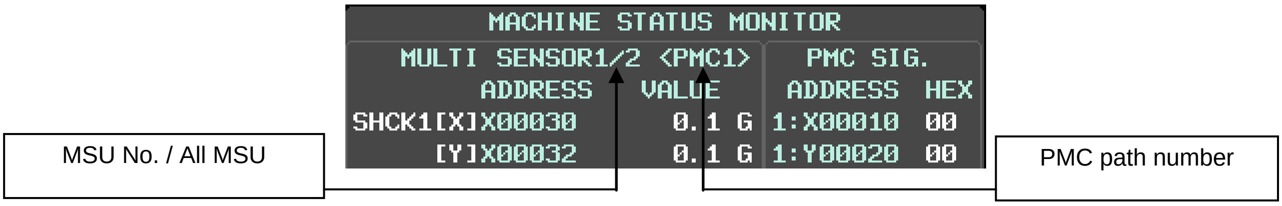
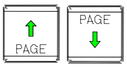


Fig. 19.1.7.2 (d) Multi PMC System and plural Multi-Sensor Units

Sensor Item

The following sensor items are displayed.

(1) Shock Sensor (Direction of X, Y, Z) : 2CH
(2) Analog Input : 2CH
(3) Thermo Sensor : 8CH

When all sensors cannot be displayed in one screen, the page can be switched by pushing page change key .

Address

The address corresponding to each sensor is displayed.

Value

The sensor value that each sensor outputs is displayed.

Unit

The unit corresponding to each sensor is displayed.

PMC Signal Title

Refer to "Monitoring of PMC signals".

PMC Signal Address

Refer to "Monitoring of PMC signals".

PMC Signal Value

Refer to "Monitoring of PMC signals".

Operation

If the soft key [(OPRT)] is pressed, the following soft keys are displayed.

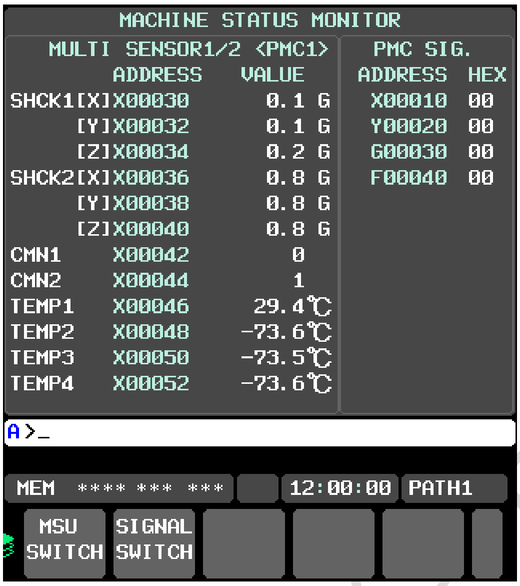


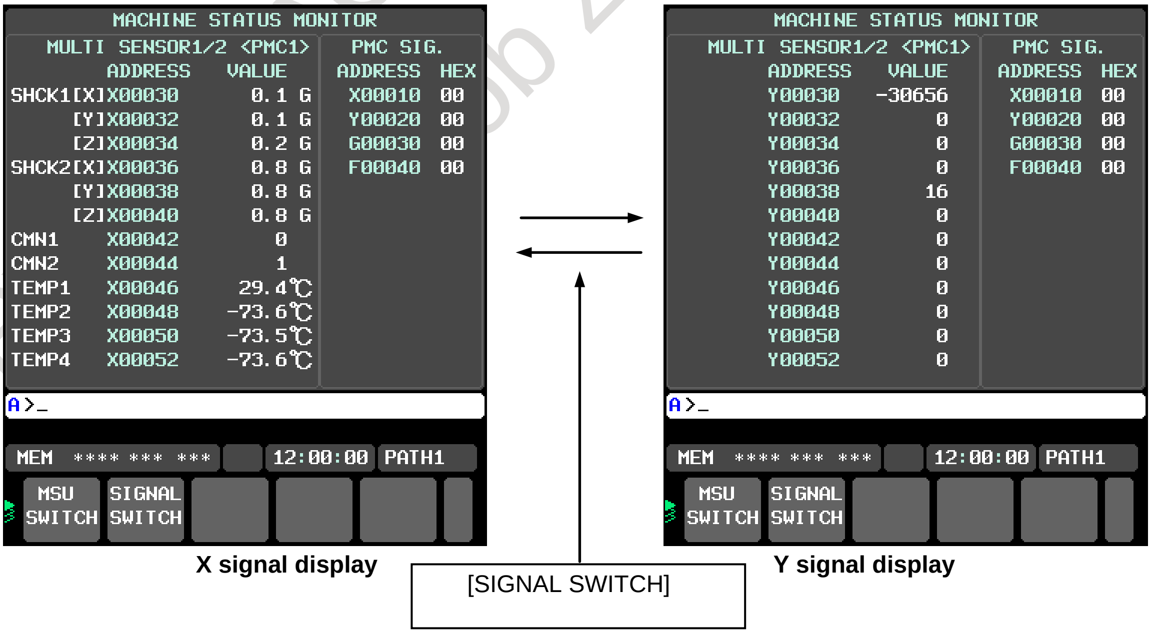
Fig. 19.1.7.2 (e) soft key [(OPRT)]

Soft Key [MSU SWITCH]

This soft key is displayed when the plural Multi-Sensor Units are connected.
When this soft key is pressed, the displayed Multi-Sensor Units can be switched.
For example, when four Multi-Sensor Units are connected, whenever this soft key is pushed, the displayed Multi-Sensor Unit changes as MSU No.1 to 2 to 3 to 4 to 1…

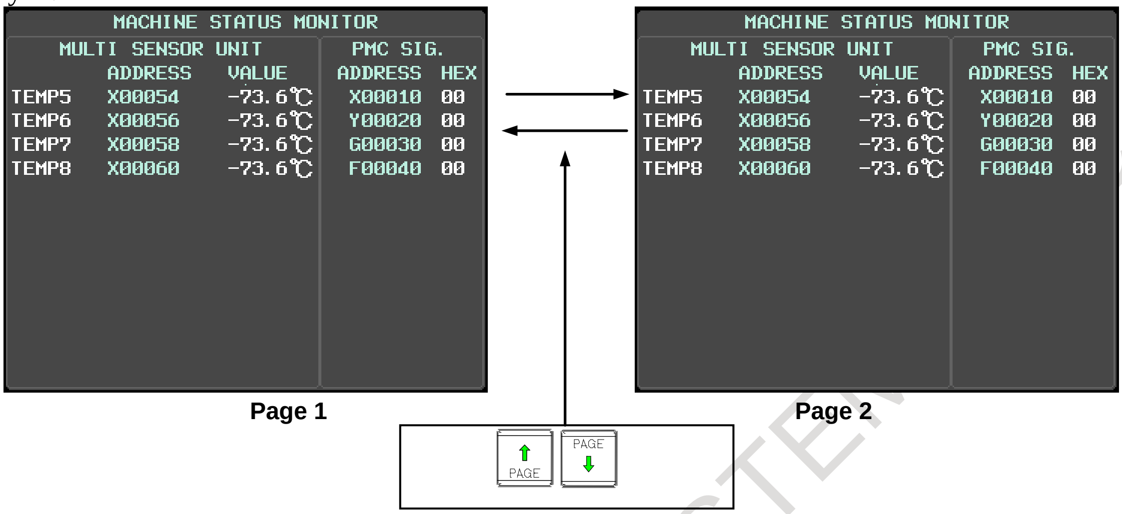
Soft Key [SIGNAL SWITCH]

When this soft key is pressed, either display of X or Y signal can be switched.



Page Change Key

When the page change key is pressed, the sensor signal information of the next page can be displayed.



19.1.8 Machine state history screen

When the machine state history save request condition is fulfilled, CNC saves the PMC signal, the state of CNC and the operation history in CNC as a history.
In the machine state history screen, these saved histories can be displayed.
As a result, for example, the machine state (coordinate, program etc.) and operation history at the time when a spindle collides can be checked and the investigation of cause of trouble and restoration are easy.

The histories can be saved up to 100.

NOTE
1. If the machine state history save request condition is fulfilled when the 100 of history data have been saved, the new history overwrites the oldest one.
2. Don't set the machine state history save request condition fulfilled while the machine state history screen is displayed. In this case, CNC Information is saved when the display of the screen changes other screen.

The state of CNC saved as a history is as follows.

Table 19.1.8 (a) Saved CNC Information List

Item	Remarks
Time	Year, Month, Day, Hour, Minute, Second
PMC Signal	4bytes
Machine Position	All Axis, All Path
Absolute Position	All Axis, All Path
Feedrate	All Path
Spindle Speed	All Axis, All Path
Program Name (O Number)	Executing program name, All Path
Block Number	Block number of executing program, All Path
Mode	All Path
Modal	M, S, T, B (Without G code) All Path
Operation History	Data of 1/20 of normal operation history. About 400 MDI keys. The number of MDI keys decreases when a history with large size such as the alarm history is saved.
Non-Save Number	Number of information not saved.

Item	Remarks
Multi-Sensor Unit Signals	Maximum 4
Save trigger	SIGNAL : signal MSUREQ ABTRQ : unexpected abnormal torque is detected SV ALM : servo alarm SP ALM : spindle alarm

The machine state history screen is composed by the following screens:
(A) Machine state history list screen
(B) Machine state history CNC data screen
(C) Machine state history operation history screen

The machine state history list screen is displayed by the following procedures.

1. Press function key .
2. Press continuous menu key [+] several times to display the soft key [MACHIN HISTORY]
3. Press the soft key [MACHIN HISTORY].
4. Press the soft key [LIST], and machine state history list screen is displayed.
5. Press the soft key [CNC DATA], and CNC data screen is displayed.
6. Press the soft key [OPERAT HISTORY] to display the operation history screen is displayed.

19.1.8.1 Machine State History List Screen

The machine state history list screen is composed by the following area.
(1) Common area
(2) History list area
(3) Detail data area

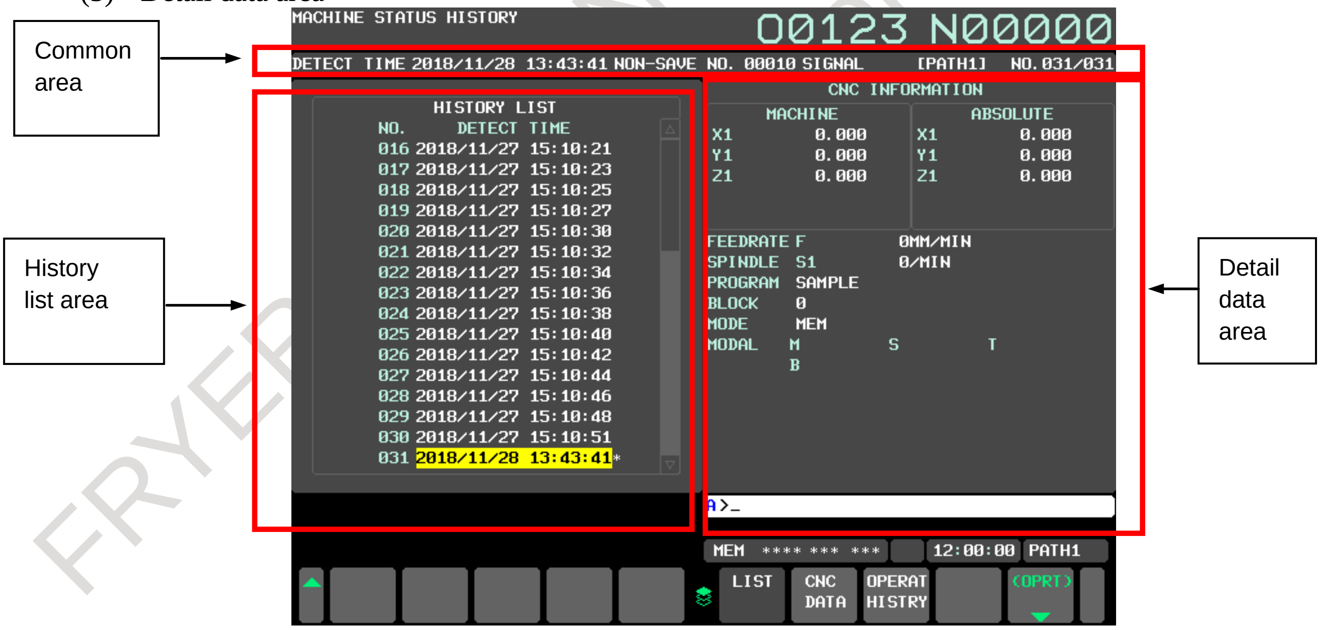


Fig. 19.1.8.1 (a) Machine State History List Screen

In common area, common information such as the detect time and non-save number of the selection history number is displayed.
In history list area, the detect time of all the histories saved in CNC is displayed in the list.
In detail data area, the detail data of the selection history number is displayed.
The following kind of detail data can be displayed.
(a) Signal information
(b) CNC information
(c) Operation history information

These kinds can be selected by cursor key [←] or [→].

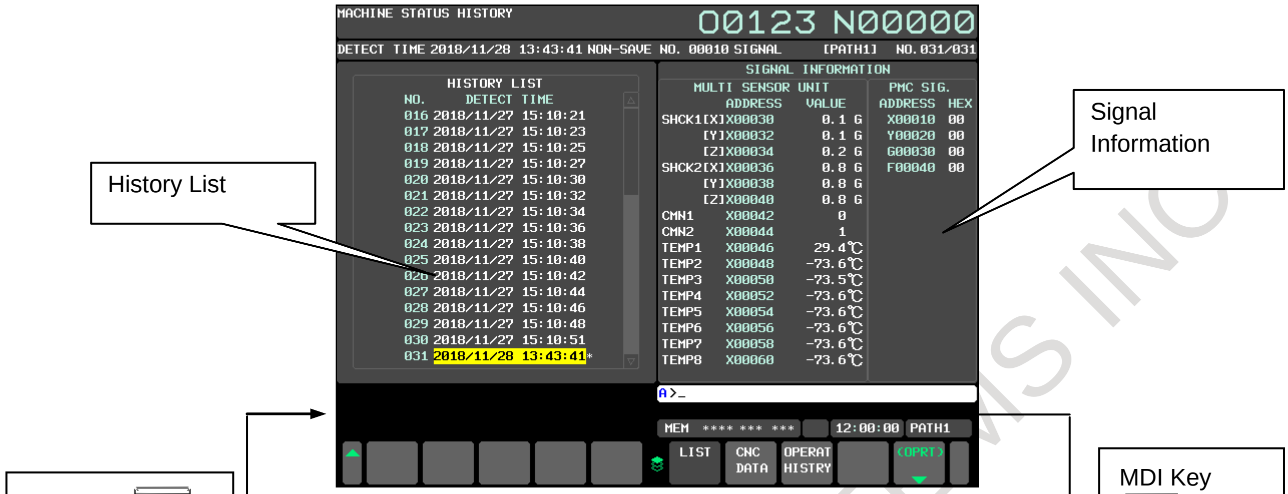


Fig. 19.1.8.1 (b) Machine State History List Screen (List + Signal)

Fig. 19.1.8.1 (c) Machine State History List Screen (List + CNC)

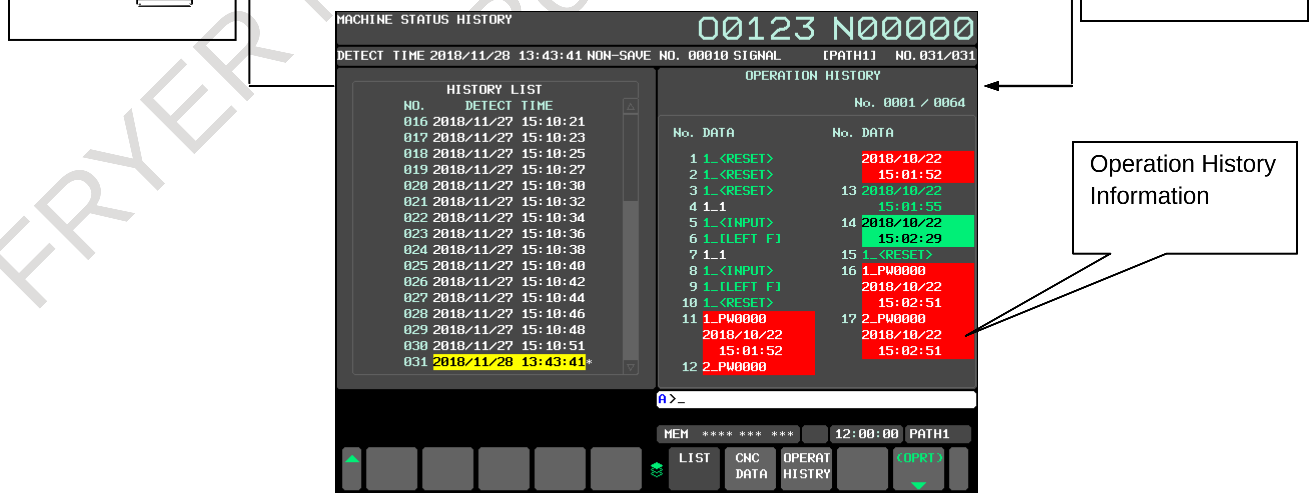


Fig. 19.1.8.1 (d) Machine State History List Screen (List + Operation History)

Common Information

Common information is displayed in the upper part of the screen.

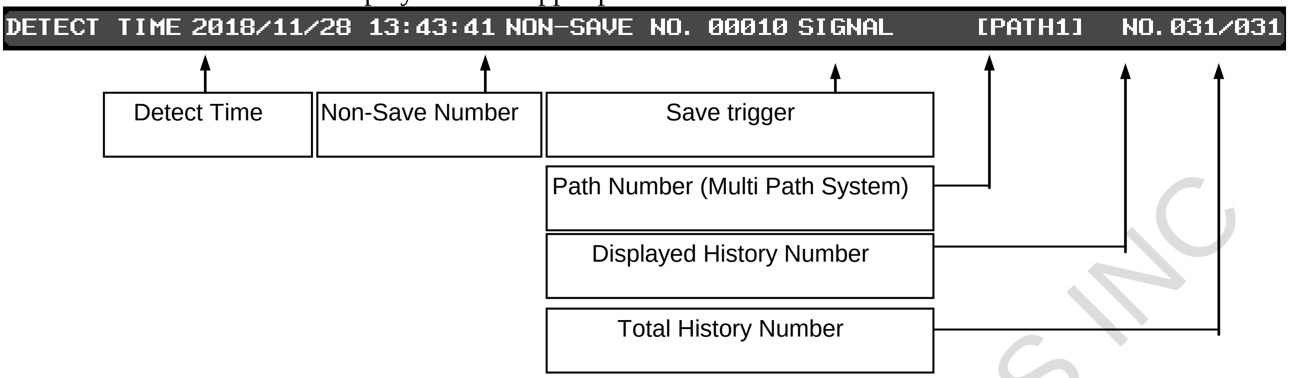


Fig. 19.1.8.1 (e) Common Area

(1) Displayed history number
The selected history number is displayed. A history number is smaller, a data is older.

(2) Total history number
The total number of machine state histories saved in CNC is displayed.

(3) Detect time
The time that the machine state history save request signal MSUSRQ<G0534.6> becomes “1” and the history was saved is displayed.

(4) Path number
In the case of multi path system, the path number of the displayed history is displayed.

(5) Non-Save Number
Non-Save Number is displayed.
The number of history that was not saved due to the number limit is displayed.
When Non-Save Number is 0, it is not displayed on screen.
The maximum value of Non-Save Number is 65535.

(6) Save trigger
Trigger when saving history is displayed.

History List information

The history list information is displayed in the area in the left half of the machine state history list screen.
History number and detect time are displayed.
If Non-Save Number is not 0, “*” is displayed.

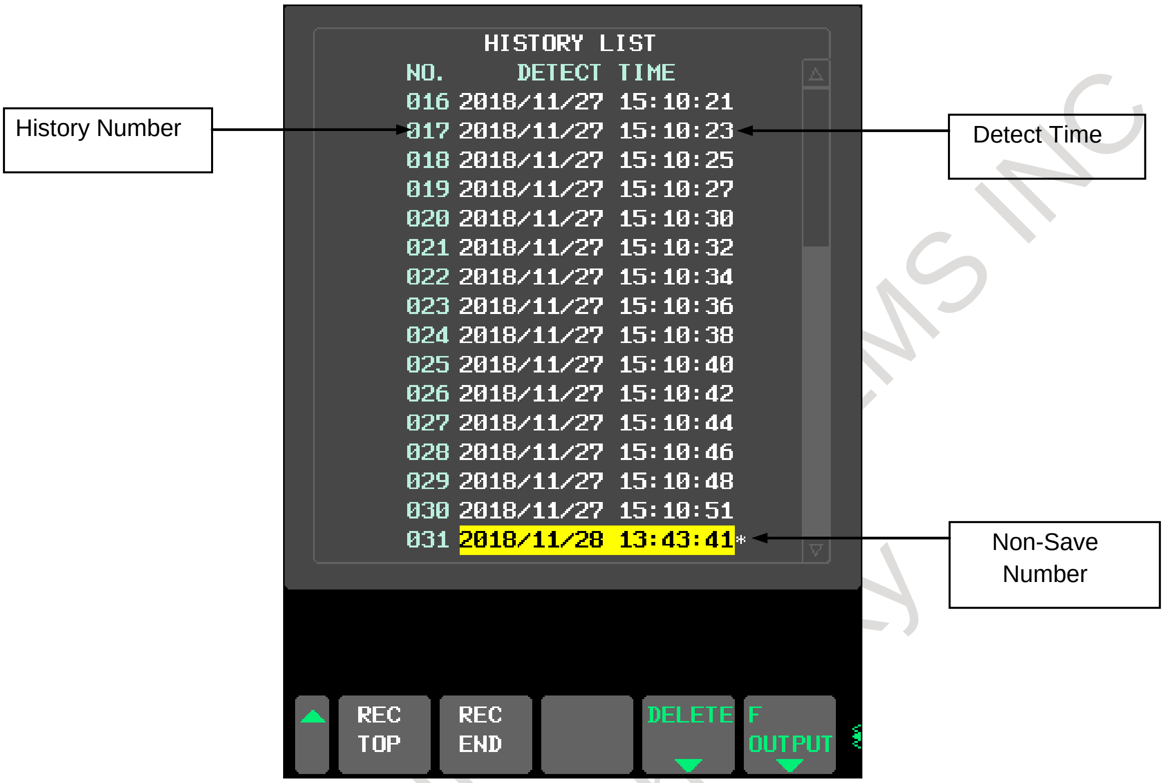
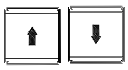
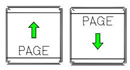


Fig. 19.1.8.1 (f) History List Area

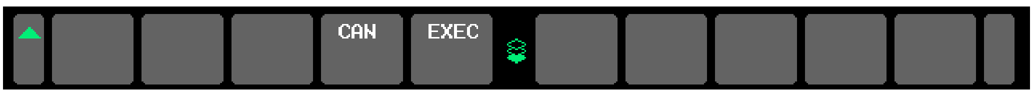
Operation in History List Area

(1) Cursor Key and Page Change Key

The cursor is moved up and down by the cursor key  and page change key , and the displayed history number can be changed.

(2) Soft Key [REC TOP] and [REC END]
If the soft key [REC TOP] is pressed, the oldest history number is displayed.
If the soft key [REC END] is pressed, the latest history number is displayed.

(3) Soft Key [F OUTPUT]
When the emergency stop state or EDIT mode is entered and the soft key [F OUTPUT] is pressed, the following soft keys are displayed.



When any file name is input and the soft key [EXEC] is pressed, all the histories saved in CNC are output.
If no name is input and the soft key [EXEC] is pressed, file name is “MSRHSTRY.TXT”

Signal Information

The signal information is displayed in the area in the right half of the machine state history list screen. The signal information of selected number history is displayed.

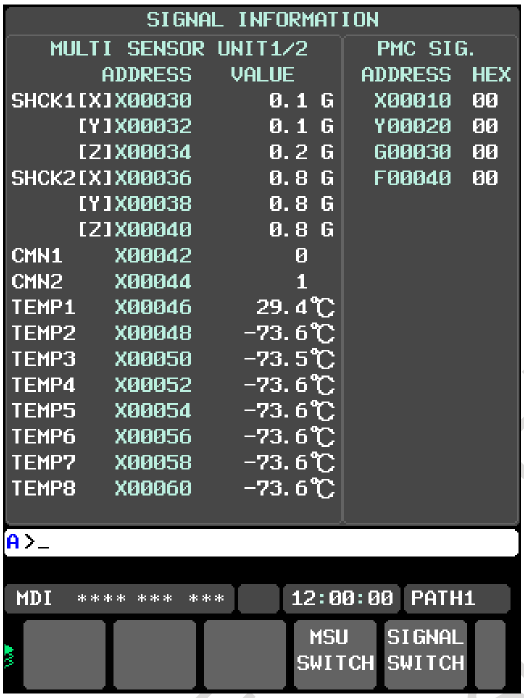


Fig. 19.1.8.1 (g) Signal Information

The content of the display and the operation are similar to the machine state monitoring screen. Refer to “Monitoring of PMC signals” and “With Multi-Sensor Unit”.

CNC Information

The CNC information is displayed in the area in the right half of the machine state history list screen. The CNC information of selected number history is displayed.

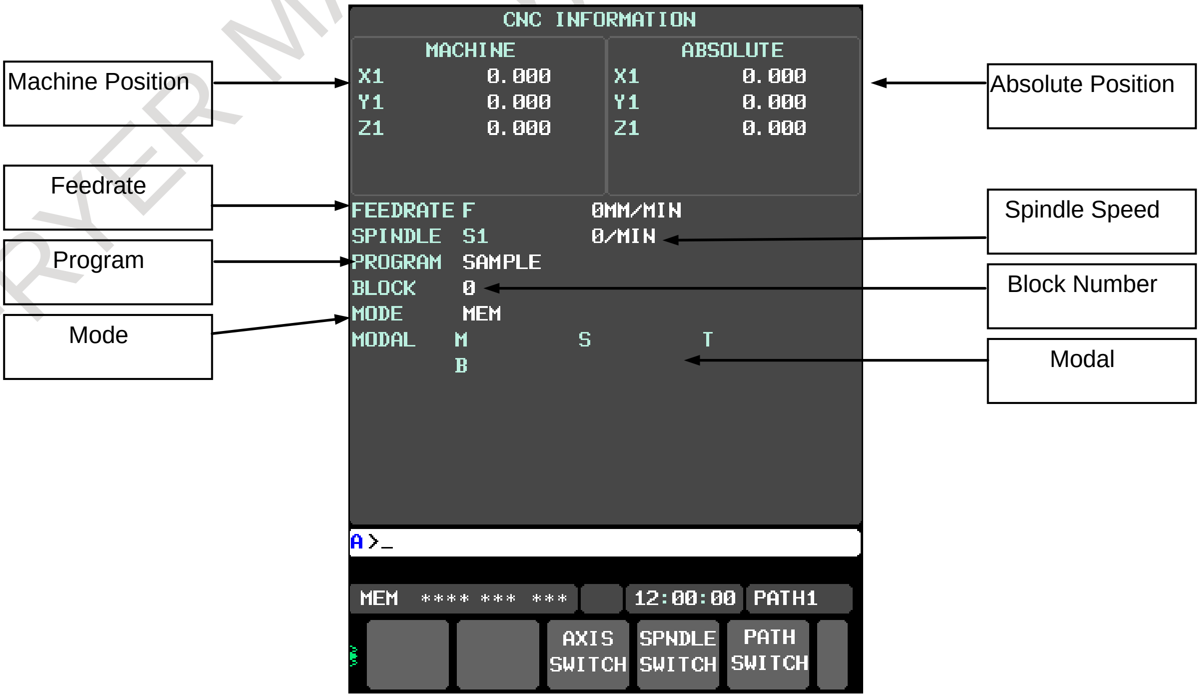


Fig. 19.1.8.1 (h) CNC Information

(1) Machine Position
Machine position is displayed.
When the number of control axis is six or more, if the soft key [AXIS SWITCH] is pressed then the axis since the sixth axis is displayed.

(2) Absolute Position
Absolute position is displayed.
When the number of control axis is six or more, if the soft key [AXIS SWITCH] is pressed then the axis since the sixth axis is displayed.

(3) Feedrate
Feedrate is displayed.

(4) Spindle Speed
Spindle speed is displayed.
When the number of spindle is two or more, if the soft key [SPINDLE SWITCH] is pressed then the spindle since the second spindle is displayed.

(5) Mode
Mode is displayed.

(6) Modal
Modal is displayed.
The modal that can be displayed is M, S, T, and B.

Operation in CNC Information

(1) Soft Key [AXIS SWITCH]
When the number of control axis is six or more, the soft key [AXIS SWITCH] is displayed.
If this soft key is pressed, the positions of the sixth and after axes are displayed in the machine and absolute coordinates.

(2) Soft Key [SPINDLE SWITCH]
When the number of spindle is two or more, the soft key [SPINDLE SWITCH] is displayed.
If this soft key is pressed, the speed of the second and after spindles is displayed.

(3) Soft Key [PATH SWITCH]
When the number of paths are two or more, the soft key [PATH SWITCH] is displayed.
If this soft key is pressed, the displayed path number is changed.

Operation History Information

The operation history information is displayed in the right half of the machine state history list screen.
The operation history until the time that the selected history was saved is displayed.
The operation history is saved about 400 MDI keys.

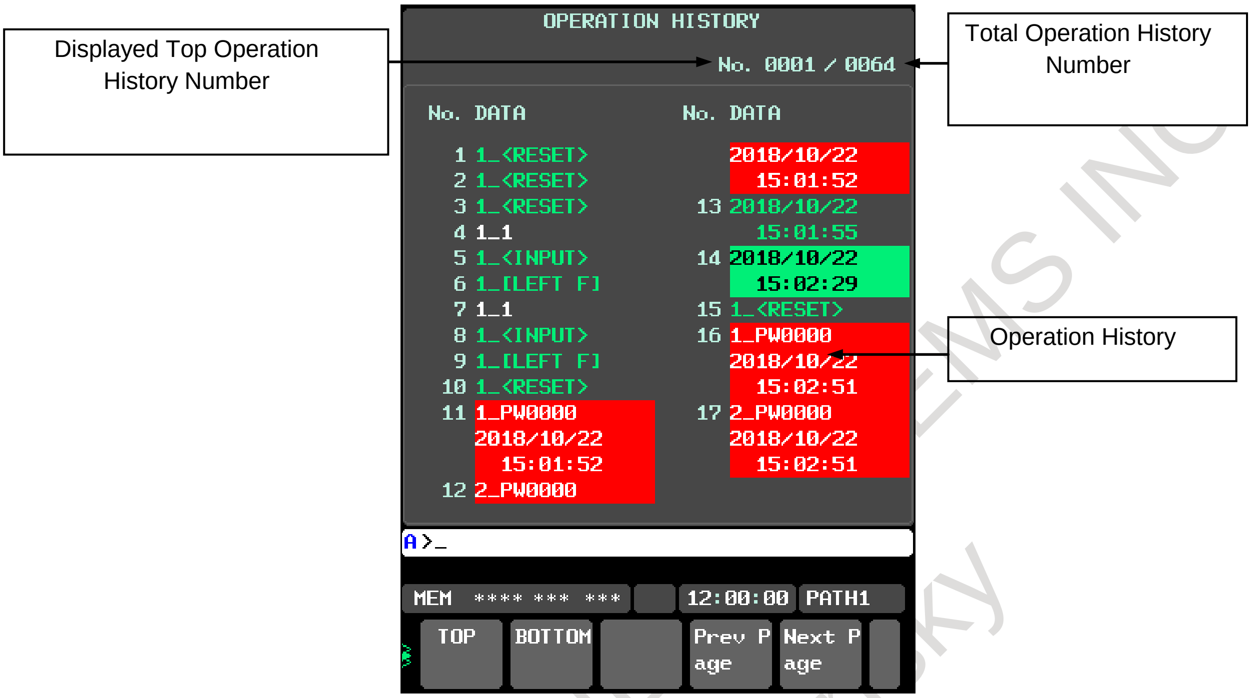


Fig. 19.1.8.1 (i) Operation History Information

Operation in Operation History Information

(1) Soft Key [TOP]
The oldest operation history is displayed.

(2) Soft Key [BOTTOM]
The newest operation history is displayed.

(3) Soft Key [Prev Page]
The previous page is displayed.

(4) Soft Key [Next Page]
The next page is displayed.

19.1.8.2 Machine State History CNC Data Screen

In the machine state history CNC data screen, the common information, the signal information and CNC information are displayed together.
In each machine state history screen, If the soft key [CNC DATA] is pressed, the machine state history CNC data screen is displayed.

Fig. 19.1.8.2 (a) CNC Data Screen

Refer to "Machine State History List Screen" for each information.

19.1.8.3 Machine State History Operation History Screen

In the machine state history operation history screen, the common information and operation history information are displayed together.
If the soft key [OPERAT HISTRY] is pressed, the machine state history operation history screen is displayed.
The operation history information is displayed in the full screen.

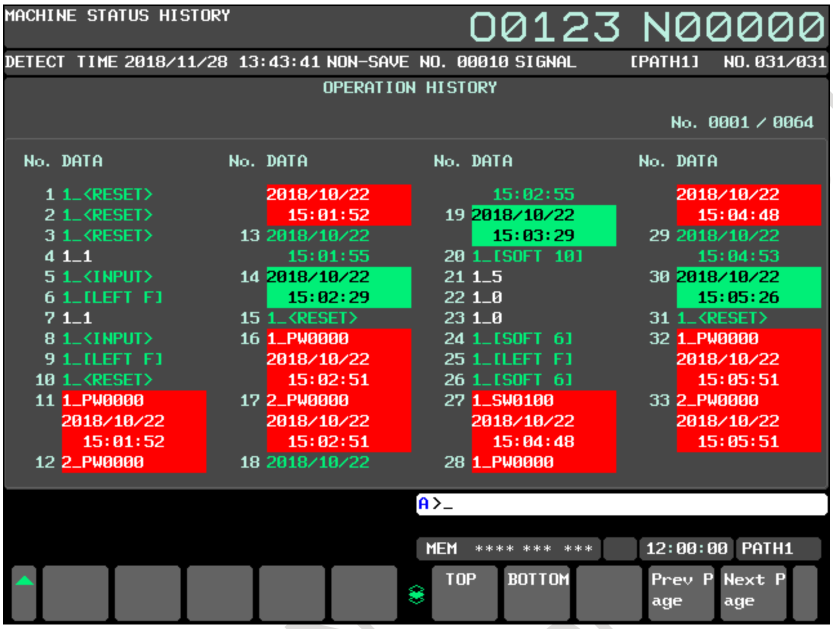


Fig. 19.1.8.3 (a) Operation History Screen

Refer to “Machine State History List Screen” for the displayed contents and the operation.

19.1.8.4 Output CNC Information

When the soft key [F OUTPUT] is pressed in the machine state history list screen, all history saved in CNC is output.
The composition of the output file is as follows.

HEADER
HISTORY DATA 1
HISTORY DATA 2
:
HISTORY DATA N

Fig. 19.1.8.4 (a) The composition of the output file

Header

In the header, total history number and file information is output. The file information is for maintenance.

NUMBER OF HISTORY : 010 ← Total history number
FILE INFORMATION : 1-100-10-9 ← File Information (For maintenance)

History Data

In the history data, history number, detect time, Non-Save Number, save trigger, Multi-Sensor Unit data(All MSU), PMC signal, CNC data(All Path), operation history and auxiliary information are output.

(1) History number, detect time and Non-Save Number.
The history number and the detect time are output as follows.
[HISTORY NO.001]-------------------------------- ← History Number
DETECT TIME : 2010/03/10 21:15:07 ← Detect Time
NON-SAVE NO. : 00025 ← Non-Save Number
SAVE TRIGGER: ABTRQ ← Save Trigger

(2) Multi-Sensor Unit Data

The Multi-Sensor Unit data is output as follows. In the case of the plural Multi-Sensor Units, all Multi-Sensor Unit data are output.

```
MULTI SENSOR UNIT DATA :
    <MULTI SENSOR UNIT NO.1>
        SHOCK1 X(X00000) :      0.0 [G]
        SHOCK1 Y(X00002) :      0.0 [G]
        SHOCK1 Z(X00004) :     29.9 [G]
        SHOCK2 X(X00006) :      1.9 [G]
        SHOCK2 Y(X00008) :      0.0 [G]
        SHOCK2 Z(X00010) :      0.0 [G]
        COMMON1 (X00012) :        0 [-]
        COMMON2 (X00014) :        0 [-]
        TEMP1    (X00016) :     0.0 [C]
        TEMP2    (X00018) :     0.0 [C]
        TEMP3    (X00020) :     0.0 [C]
        TEMP4    (X00022) :     0.0 [C]
        TEMP5    (X00024) :     0.0 [C]
        TEMP6    (X00026) :     0.0 [C]
        TEMP7    (X00028) :     0.0 [C]
        TEMP8    (X00030) :     0.0 [C]
                (Y00000) : -30656
                (Y00002) :      0
                (Y00004) :      0
                (Y00006) :      0
                (Y00008) :     16
                (Y00010) :      0
                (Y00012) :      0
                (Y00014) :      0
                (Y00016) :      0
                (Y00018) :      0
                (Y00020) :      0
                (Y00022) :      0
                (Y00024) :      0
                (Y00026) :      0
                (Y00028) :      0
                (Y00030) :      0
```

(3) PMC Signal

The PMC signal is output as follows.

```
Single PMC System
PMC SIGNAL DATA :
    X00000 0x3F
    X00001 0x04
    Y00000 0x00
    Y00008 0xF0
Multi PMC System:
PMC SIGNAL DATA :
    1:X00000 0x3F
    1:X00001 0x04
    1:Y00000 0x00
    1:Y00008 0xF0
```

(4) CNC Data

The CNC data is output as follows. In the case of multi path system all path data are output.

```
CNC DATA :
    <<MACHINE>>
        X           0.000
        Y           0.000
        Z           0.000
    <<ABSOLUTE>>
        X           0.000
        Y           0.000
        Z           0.000
    <<FEED RATE>>
        F               0
    <<SPINDLE SPEED>>
        S               0
    <<PROGRAM>>
        //CNC_MEM/USER/PATH1/O123
    <<BLOCK>>
        0
    <<MODE>>
        MEM
    <<MODAL>>
        M               0
        S               0
        T               0
        B               0
```

(5) Operation History

The operation history is output as follows.

```
OPERATION HISTORY :
MDI 01_[RIGHT F] 12:02:13
MDI 01_[RIGHT F] 12:02:14
MDI 01_[RIGHT F] 12:02:15
MDI 01_[RIGHT F] 12:02:15
MDI 01_[SOFT HF7] 12:02:16
MDI 01_<Cursor Down> 12:02:26
MDI 01_<Cursor Down> 12:02:26
 :
```

(6) Auxiliary Information

The auxiliary information is output. This information is for maintenance.

```
AUX INFO :
0-0
```

Reading of CNC information by FOCAS2/C language executor

Information saved in CNC can be read by the following functions of the FOCAS2 library and the C language library of C language executor.

Table 19.1.8.4 (a)

Function Name	Function
cnc_upstart4	Beginning of NC data read (4)
cnc_upload4	NC data read (4)
cnc_upend4	End of NC data read (4)

When cnc_upstart4 is called specifying 78 for argument type, CNC information can be read.
A basic procedure of reading CNC information by cnc_upstart4, cnc_upload4, and cnc_upend4 function is as shown in fig.19.1.8.4 (b).

Please refer to “10.The function specification” of FANUC FOCAS1/2 library specifications (FWLIB32.HTM) included in “FOCAS1/2 library disk” for details of cnc_upstart4, cnc_upload4, and cnc_upend4 function of the FOCAS2 library.

Please refer to “the CNC/PMC window library” of C language Executor programming manual (B-63943EN-3) for details of cnc_upstart4, the cnc_upload4, and cnc_upend4 function of the C language library of C language executor.

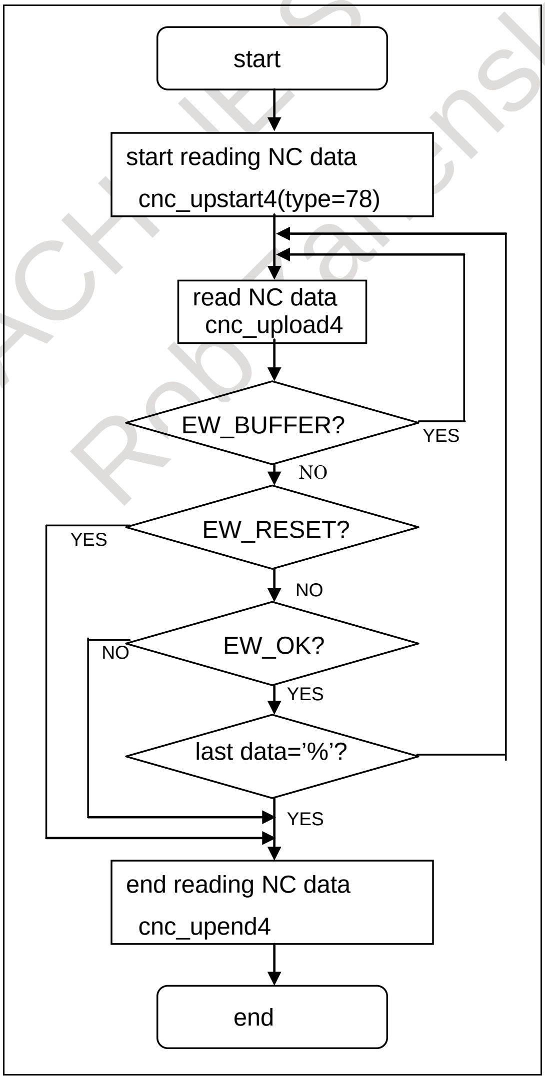


Fig. 19.1.8.4 (b) Reading data

19.1.9 Caution

To save CNC information for eight or more days, so number of saving information each day is restricted. The history can be saved only up to the potential saving number. (Refer to “Restriction of saving”)
So, in the ladder program, please judge trouble as accurately as possible.

Do not save CNC information at the time when trouble does not occur in order to decrease the load of CNC.

At shipment of CNC, Potential Saving Number is 30.

CNC information might not be able to be saved correctly immediately before the power failure. So in the system that can detect power failure, don’t save the CNC information when the power failure occurs.

19.2 SERVO/SPINDLE WAVEFORM DATA OUTPUT FUNCTION

19.2.1 Outline

When something trouble occurs with machine, servos and spindles waveform data before and after the trouble is automatically output to Data Server, which is sampled by Trouble diagnosis function.
Since long-termed data is automatically saved, it is easier to detect the cause of the trouble.

CNC always samples waveform data.
If something trouble occurs with machine, CNC detects the trouble through alarms or signals and output waveform data before and after the trouble to Data Server automatically (Fig.19.2.1(a)).

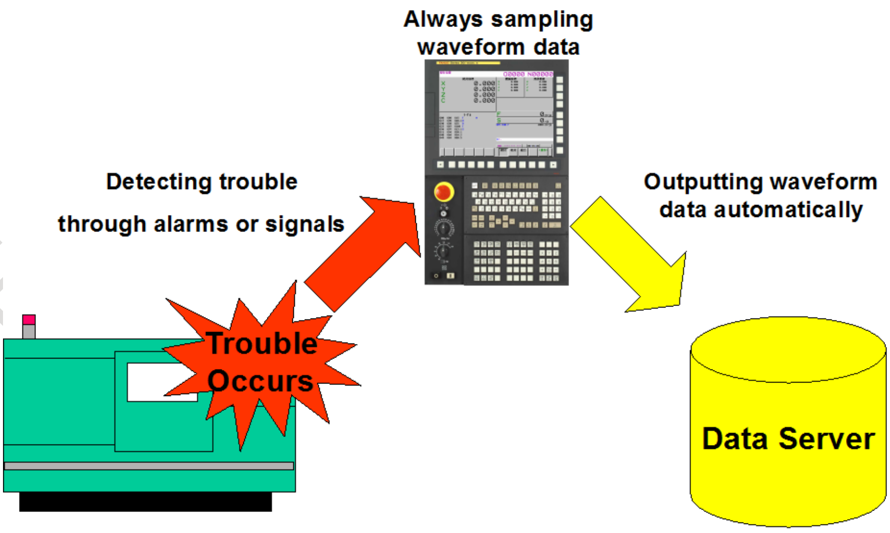


Fig. 19.2.1 (a) Outline of Servo / spindle waveform data output function

> **NOTE**
> This function requires Data Server function.

19.2.2 Waveform Data

This function can output several kinds of data listed in tables below.
Each data can be sampled for up to 40 seconds summing up before and after trouble.

Table 19.2.2 (a) Data list for servo motors

Data (Unit)	
Accumulated command pulse	(pulse)
Accumulated feedback pulse	(pulse)
Position error	(pulse)
Reference counter	(pulse)
Actual speed	(1/min)
Torque command	(%)
Head simulation	(%)
Motor current	(A)
Disturbance level	(%)
Arbitrary data	
Effective current	(%)
AMR data	
Arbitrary data 2	

Table 19.2.2 (b) Data list for spindle motors

Data (Unit)	
Actual motor speed	(1/min)
Load meter	(%)
Position error	(pulse)
Torque command	(%)
Motor current	(A)

Whole data of all axes can be output, though output data can be selected by parameters.

> **NOTE**
> The upper limit of sampling time cannot be extended even if output data is reduced.

19.2.3 Sampling

It is necessary to always sample waveform data in order to use this function.
Trouble diagnosis data monitoring signal TDDM<G0589.0> starts the sampling.
Trouble diagnosis data monitoring signal clears latched data of Trouble diagnosis function and changes the state of the function from “LATCHED” to “SAMPLING”.

Output of waveform data to Data Server starts when sampling after trouble finishes, which is specified by parameter No.24320.
It is necessary to start sampling again by trouble diagnosis data monitoring signal after outputting of waveform data.
Weather the output finishes or not is indicated by Waveform data transfer signal WVTRNS<F0578.0>.

Restriction

When this function is effective, the state of Trouble diagnosis function, “SAMPLING” or “LATCHED”, is common for all paths, though the state can be different for each path in ordinary Trouble diagnosis function. Therefore, Trouble diagnosis data monitoring signal starts sampling of all paths simultaneously and trouble in some path stops sampling of all paths.

NOTE
1. Latched data of Trouble diagnosis function and waveform data sampled by this function is cleared when sampling restarts after trouble.
2. Though Trouble diagnosis data monitoring signal is for each path, this function uses only the signal of 1st path.
3. Trouble diagnosis data monitoring signal is ignored when SV or SP alarm occurs in some path. Start sampling by Trouble diagnosis data monitoring signal after confirming that no SV or SP alarm occurs in all paths.
4. The state of sampling (the state of Trouble diagnosis function) does not change even if power is turned off and on. Therefore, it is necessary to start sampling after power is turned on if sampling was stopped (the state of Trouble diagnosis function was "LATCHED") before power was turned off. Oppositely, sampling is started automatically after power is turned on if sampling was active (the state of Trouble diagnosis function was "SAMPLING") before power was turned off.

19.2.4 Detect Trouble

SV or SP alarm changes the state of Trouble diagnosis function into "LATCHED" when the state is "SAMPLING". At the same time, CNC regards that something trouble occurs with machine and the sequence of outputting waveform data of servos and spindles to Data Server is started.

This function makes it possible to detect trouble by changing Trouble diagnosis data latching signal TDDL<G0589.1> from "0" to "1" as well as SV and SP alarms. Weather trouble is detected by Trouble diagnosis data latching signal or not is specified by bit 1 (SPT) of parameter No.24318.

NOTE
1. Though Trouble diagnosis data latching signal is for each path, this function uses the signal of 1st path.
2. Data of Trouble diagnosis function is latched and the state changes from "SAMPLING" to "LATCHED" when trouble is detected by Trouble diagnosis data latching signal.
3. Sampling continues for a while after trouble is detected, which can be specified by parameter No.24320.
4. Trouble is not detected and this function is not active when the state of Trouble diagnosis function is "LATCHED".

19.2.5 Sequence when Trouble Occurs

Servos and spindles waveform data is sampled with constant period and, if trouble occurs, data before and after the trouble specified by parameter Nos.24319 and 24320 is output to Data Server. The output is started automatically if sampling after the trouble finishes, which is specified by parameter No.24320.
Waveform data transfer signal WVTRNS<F0578.0> is "1" between the occurrence of the trouble and the end of the output.
Sequence when trouble occurs is shown in Fig. 19.2.5 (a) and Fig. 19.2.5 (b) below.

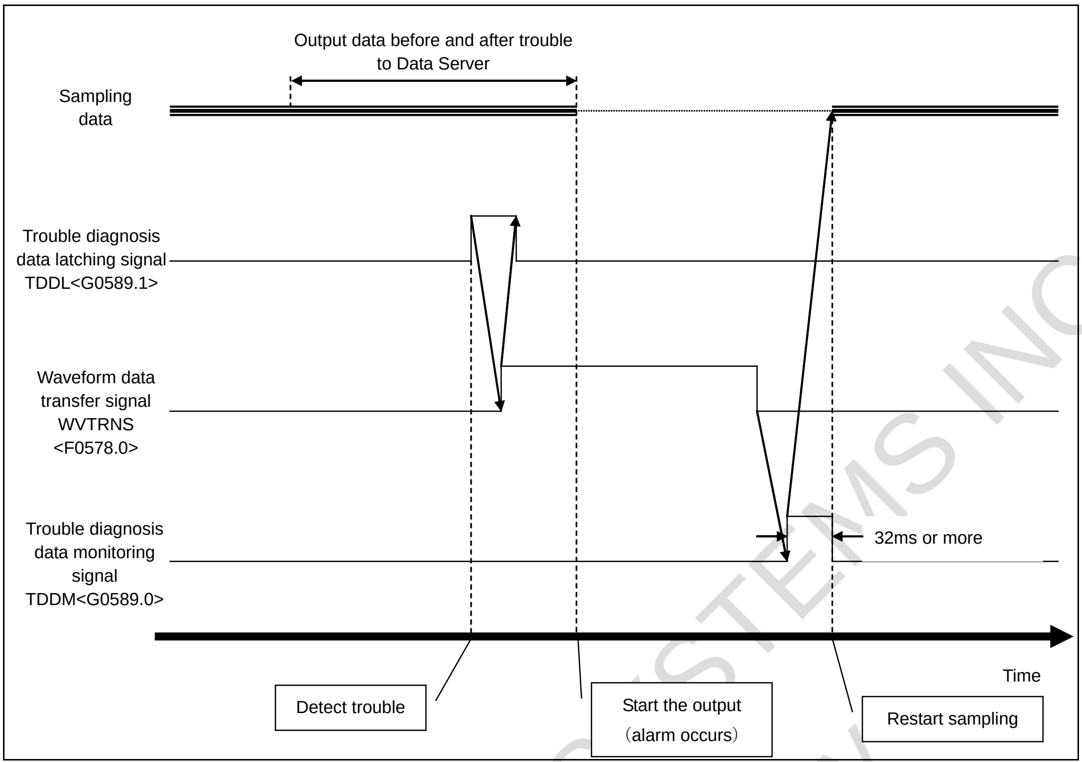


Fig. 19.2.5 (a) Sequence when trouble occurs (when the trouble is detected by the signal)

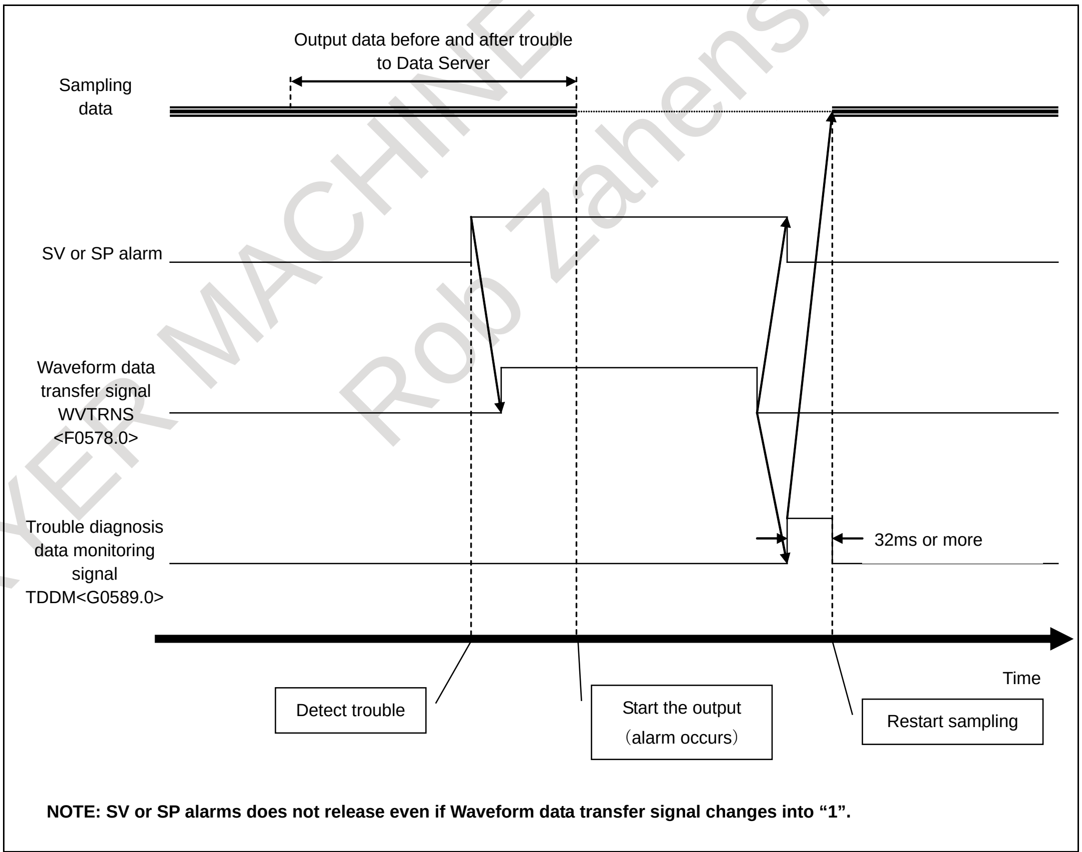


Fig. 19.2.5 (b) Sequence when trouble occurs (when the trouble is detected by SV or SP alarm)

> **NOTE**
> 1 Make it impossible to turn off power during Waveform data transfer signal is "1" in order to output data normally.
> 2 Sampling cannot be started when Waveform data transfer signal is "1" or SV/SP alarm occurs. In addition, Trouble diagnosis function and this function are not effective in such cases.

19.2.6 Data Output

Output of waveform data to Data Server is started automatically if trouble is detected and sampling after the trouble finishes, which is specified by parameter No.24320.
"OUTPUT" is blinking at Status bar during the output.
The data is output to "SVSPWAVE" folder below Data Folder on Data Server regardless of I/O devices specified in parameter Nos.20-23.
Make all paths in alarm states before the output, which is required to start the output to Data Server (See 0 in NOTE below).

Waveform data transfer failure signal WVTF<F0578.1> is "1" if the output fails because of, for examples, no data Folder existing on Data Server, memory shortage, not in alarm states, and so on. After the cause of failure is resolved while WVTF<F0578.1> is "1", setting Waveform data re-output signal WVROP<G0589.2> from "0" to "1" restarts the output in such cases.
Waveform data transfer failure signal is "0" in the following cases:
1) When the re-output finishes normally.
2) When the next sampling is started.
3) When power is turned off.
Sequence of normal output is shown in Fig. 19.2.6 (a) and Fig. 19.2.6 (b), and that of output failure is shown in Fig. 19.2.6 (c) and Fig. 19.2.6 (d).

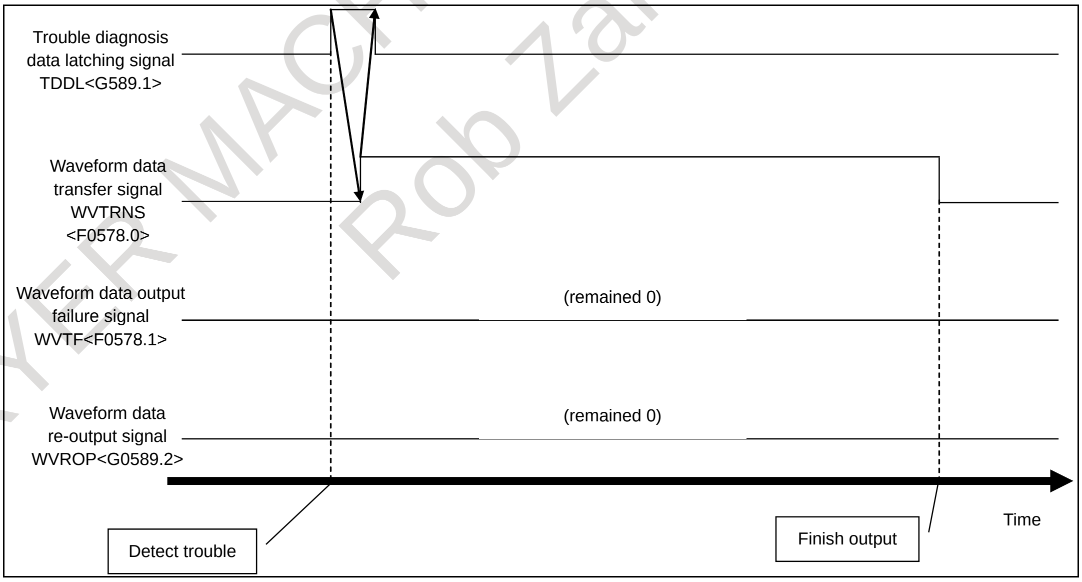


Fig. 19.2.6 (a) Sequence of normal output (when the trouble is detected by the signal)

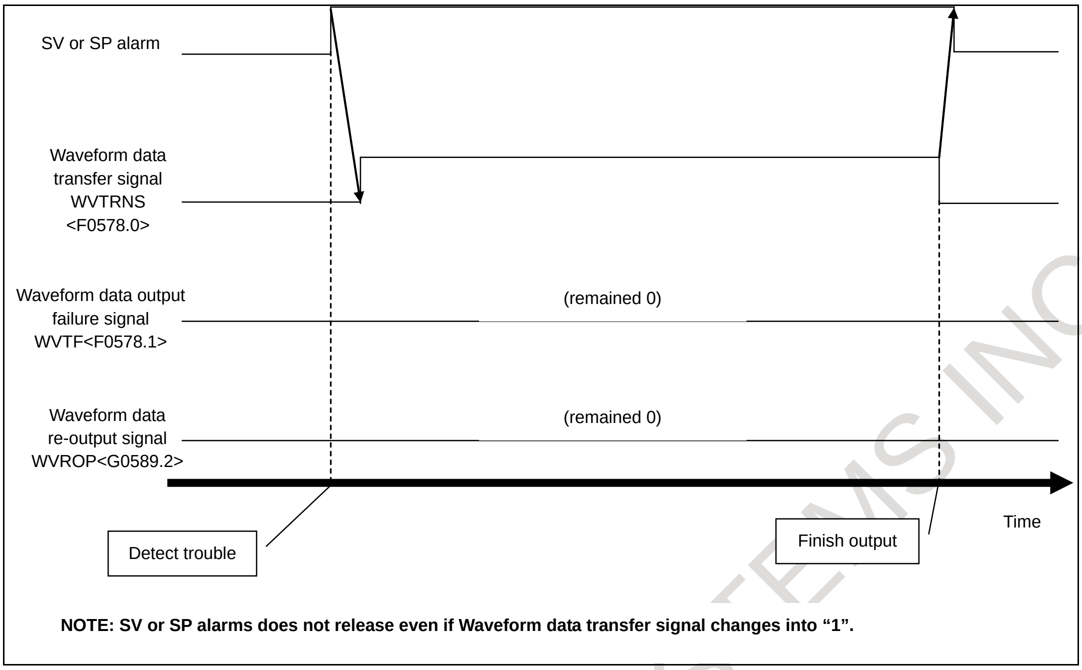


Fig. 19.2.6 (b) Sequence of normal output (when the trouble is detected by SV or SP alarm)

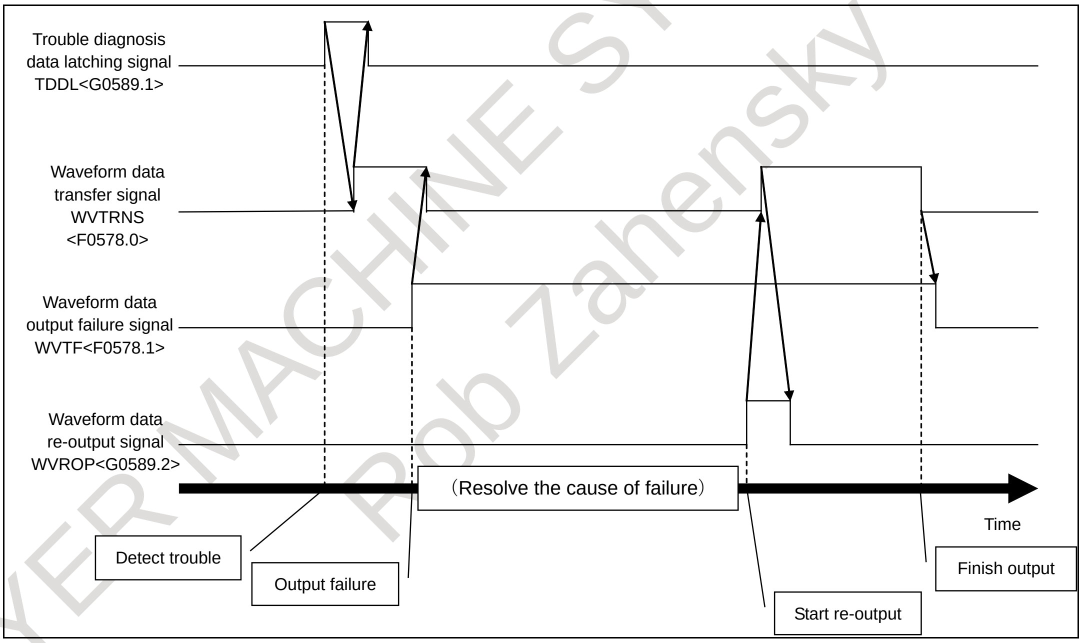


Fig. 19.2.6 (c) Sequence when output fails (when the trouble is detected by the signal)

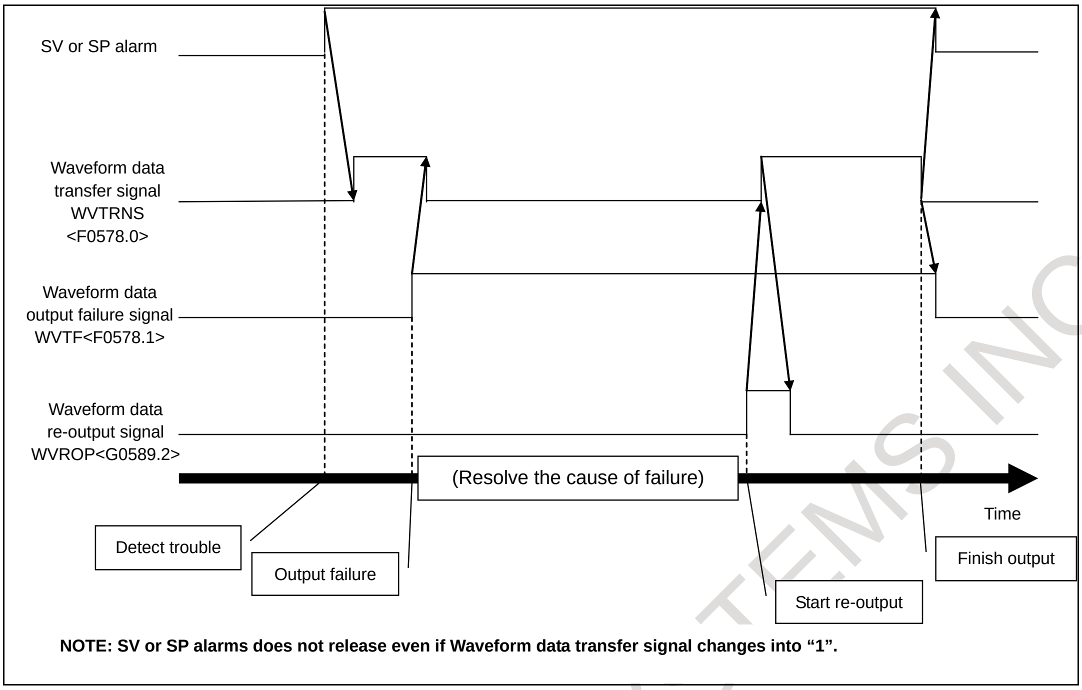


Fig. 19.2.6 (d) Sequence when output fails (when the trouble is detected by SV or SP alarm)

NOTE

1. Make all paths in alarm states by external alarms before the output starts if trouble is detected by Trouble diagnosis data latching signal (TDDL).
2. Remain alarm states during the output.
3. Output of waveform data cannot be stopped by reset operations.
4. It is impossible to output other data by screen operations during the data output. Warning message "THE MACHINE IS UNDER USE" is displayed if an output operation is made during the data output.
5. Screen hard copy cannot be used during the output. BG alarm "HARD COPY REJECTED DURING OUTPUT" occurs if screen hard copy is requested by a MDI operation or a signal. In the same way, EW_REJECT is returned if screen hard copy is requested from a FOCAS2 function.
6. Operations listed below are accepted after the output finishes if they are operated during the data output:
   - Changing the active path (only for display)
   - Saving alarm history (Time when the save is done is saved as the occurrence time of the alarm.)
   - Writing PMC data on FROM
   - Setting Memory protection signal KEYP<G0046.0>
   - Changing displayed language
7. It takes several 10 seconds or a few minutes to output whole data.
8. Re-output makes a new file involving whole data. The old defective file is overwritten by the new file.
9. Reserve enough area on Data Server to output waveform data by transferring files in "SVSPWAVE" folder to host computers etc. regularly because waveform data is always output automatically if trouble is detected.
   Diagnosis No.4600 will help the data transfer.
10. Specify ratio of capacity available for data folder and number of registered files among total capacity and the maximum number of registered files of CF card mounted on data server in parameter No.24315.

19.2.7 Output File Format

Waveform data is output in CSV format.

Filename

Output filenames are in the format “SVSPW_yyyymmddHHMMSS.CSV”.
The part of “yyyymmddHHMMSS” means date when trouble occurs.

- yyyy: year (2000-2999)
- mm: month (01-12)
- dd: day (01-31)
- HH: hour (00-23)
- MM: minute (00-59)
- SS: second (00-59)

File format

File format is as shown below.

%							

Wave Data of Trouble Diagnosis							←Title

Trouble Occurs: November 11 2011 at 14:00:00 [= 30000ms in data]							←Time when trouble occurs
Trigger : Signal (G589.1) ON							←How to detect trouble

	SV1	SV1	…	SV2	SV2	…	←Axis number
TIME	COM. PULSE	F.B. PULSE	…	COM. PULSE	F.B. PULSE	…	←Data name
[ms]	[pulse]	[pulse]	…	[pulse]	[pulse]	…	←Data unit
0	50	50	…	50	50	…	←Data
4	50	50	…	50	50	…	
8	50	50	…	50	50	…	
12	50	50	…	50	50	…	
…	…	…	…	…	…	…	
%							

19.2.8 Restriction

Sampling of this function stops when sampling of Waveform diagnosis function is active.
Note that Servos and spindles waveform data is not output if something trouble occurs during sampling of Waveform diagnosis function is active.

Signal

Trouble diagnosis data monitoring signal TDDM<Gn589.0>

[Classification] Input signal

[Function] Clears latched data of Trouble diagnosis function and changes the state from "LATCHED" to "SAMPLING". In addition, sampling of all paths starts in Servo / spindle waveform data output function.

[Operation] When this signal is set from "0" to "1", latched data of Trouble diagnosis function is cleared and the state changes from "LATCHED" to "SAMPLING". In addition, sampling of waveform data starts. Note that this signal is ignored when Wave data transfer signal WVTRNS is "1". Since this signal is also ignored when SV or SP alarm occurs, set this signal after resolving the causes of alarms.

> **NOTE**
> Only the signal of 1st path is effective when Servo / spindle waveform data output function is effective.

Trouble diagnosis data latching signal TDDL<Gn589.1>

[Classification] Input signal

[Function] Latches data of Trouble diagnosis function and changes the state from "SAMPLING" to "LATCHED". In addition, sequence of outputting waveform data starts in Servo / spindle waveform data output function.

[Operation] When this signal is set from "0" to "1", data of Trouble diagnosis function is latched and the state changes from "SAMPLING" to "LATCHED". In addition, sequence of outputting waveform data starts. Note that this signal is ignored when the state of Trouble diagnosis function is "LATCHED".

> **NOTE**
> Only the signal of 1st path is effective when Servo / spindle waveform data output function is effective.

Waveform data re-output signal WVROP<G0589.2>

[Classification] Input signal

[Function] Starts re-output of waveform data if output of servos and spindles waveform data fails because of shortage of memory and so on.

[Operation] When this signal is set from "0" to "1", re-output of waveform data is started. Start re-output by this signal after resolving causes of output failure. Note that this signal is ignored when Waveform data transfer failure signal WVTF is "0".

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn589						WVROP	TDDL	TDDM

Waveform data transfer signal WVTRNS<F0578.0>

[Classification] Output signal

[Function] Notifies that trouble was detected and that output of waveform data does not finish in Servo / spindle waveform data output function. Note that this signal is “1” even if output of waveform data does not run during sampling after the trouble.

[Operation] This signal is “1” when output of waveform data does not finish.
This signal is “0” when output of waveform data finishes.

Waveform data transfer failure signal WVTF<F0578.1>

[Classification] Output signal

[Function] Notifies that output of waveform data failed in Servo / spindle waveform data output function. Re-output of the data can be started by Waveform data re-output signal when this signal is “1”.

[Operation] This signal is “1” when output of waveform data failed.
This signal is “0” in the following cases:

1. Output of waveform data was normal.
2. Sampling of waveform data was restarted.
3. Power was turned off and on.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn578							WVTF	WVTRNS

Parameter

24315	Ratio of use of data folder of data server

[Input type] Parameter input

[Data type] Byte system

[Valid data range] 0 to 9(common to the system)

Specify ratio of capacity available for data folder and number of registered files among total capacity and the maximum number of registered files of CF card mounted on data server.

For example, when formatting is executed by setting parameter DFR to 3, while using a CF card with 2 GB capacity, in data folder it is possible to use up to about 0.6 GB, which is 30% of the total capacity, and available capacity of NC program storage area is approximately 1.4 GB.

Also, if 2047 files can be registered, the maximum number of registered files is 614 in the data folder and 1433 in NC program. If 0 is specified, the data folder cannot be available and the data folder is not created even when formatting.

When 10 or more or -1 or less is specified, it assumed to be 5.

	#7	#6	#5	#4	#3	#2	#1	#0
24318							SPT	WDT

[Input type] Parameter input

[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 WDT Servo / spindle waveform data output function is

0: Effective.

1: Not effective.

#1 SPT In Trouble diagnosis function and Servo / spindle waveform data output function, Trouble diagnosis data monitoring signal and Trouble diagnosis data latching signal are
0: Ignored.
1: Not ignored.

24319	Sampling time of waveform data before trouble

[Input type] Parameter input
[Data type] Byte
[Unit of data] sec
[Valid data range] 0 to 40

24320	Sampling time of waveform data after trouble

[Input type] Parameter input
[Data type] Byte
[Unit of data] sec
[Valid data range] 0 to 40

Set sampling times before and after trouble in Servo / spindle waveform data output function.
Specify the sampling times so that the sum of them is within 40 seconds because room for the buffer is limited. Note that data is not sampled as specified by these parameters if the sum exceeds 40 seconds.
The following is an example when the sum exceeds 40 seconds:
When 30 seconds is specified for the sampling time before trouble and 20 seconds is specified for the sampling time after trouble, waveform data of 30 seconds before trouble and of 10 seconds after trouble is obtained.
When 50 seconds is specified for the sampling time before trouble and 10 seconds is specified for the sampling time after trouble, waveform data of only 40 seconds before trouble is obtained.

	#7	#6	#5	#4	#3	#2	#1	#0
24321	MCR	HTS	TCM	ASD	REF	PER	FBP	CMP

[Input type] Parameter input
[Data type] Bit axis

#0 CMP In Servo / spindle waveform data output function, accumulated command pulse of a servo axis is
0: Output.
1: Not output.

#1 FBP In Servo / spindle waveform data output function, accumulated feedback pulse of a servo axis is
0: Output.
1: Not output.

#2 PER In Servo / spindle waveform data output function, position error of a servo axis is
0: Output.
1: Not output.

#3 REF In Servo / spindle waveform data output function, reference counter of a servo axis is
0: Output.
1: Not output.

#4 ASD In Servo / spindle waveform data output function, actual speed of a servo axis is

0: Output.
1: Not output.

#5 TCM In Servo / spindle waveform data output function, torque command of a servo axis is
0: Output.
1: Not output.

#6 HTS In Servo / spindle waveform data output function, heat simulation of a servo axis is
0: Output.
1: Not output.

#7 MCR In Servo / spindle waveform data output function, motor current of a servo axis is
0: Output.
1: Not output.

	#7	#6	#5	#4	#3	#2	#1	#0
24322				AR2	AMR	EFC	AR1	DLV

[Input type] Parameter input
[Data type] Bit axis

#0 DLV In Servo / spindle waveform data output function, disturbance level of a servo axis is
0: Output.
1: Not output.

#1 AR1 In Servo / spindle waveform data output function, arbitrary data 1 of a servo axis is
0: Output.
1: Not output.

#2 EFC In Servo / spindle waveform data output function, effective current of a servo axis is
0: Output.
1: Not output.

#3 AMR In Servo / spindle waveform data output function, AMR data of a servo axis is
0: Output.
1: Not output.

#4 AR2 In Servo / spindle waveform data output function, arbitrary data 2 of a servo axis is
0: Output.
1: Not output.

	#7	#6	#5	#4	#3	#2	#1	#0
24323				MCR	TCM	PER	LDM	SPD

[Input type] Parameter input
[Data type] Bit spindle

#0 SPD In Servo / spindle waveform data output function, actual motor speed of a spindle axis is
0: Output.
1: Not output.

#1 LDM In Servo / spindle waveform data output function, load meter of a spindle axis is
0: Output.
1: Not output.

#2 PER In Servo / spindle waveform data output function, position error of a spindle axis is

0: Output.
1: Not output.

#3 TCM In Servo / spindle waveform data output function, torque command of a spindle axis is
0: Output.
1: Not output.

#4 MCR In Servo / spindle waveform data output function, motor current of a spindle axis is
0: Output.
1: Not output.

Diagnosis

Diagnosis	4600	Counter of output of waveform data

[Data type] Byte
[Unit of data] 1 time
[Valid data range] 0-255

The state of output of servos and spindles waveform data can be displayed.

Power-on clears this value to 0 and then 1 is added when output starts. In addition, 1 is added again when output finishes normally. Therefore, odd number means that output is active and even number means that output is not active.

Note that this value changes as follows if output fails:
1 1 is added when output starts.
2 1 is subtracted when output fails.
3 1 is added when output starts again.
4 1 is added when output finishes normally.

NOTE
1 This value is not cleared even if power is turned off.
2 This value is counted up from 0 again if this value exceeds 255.

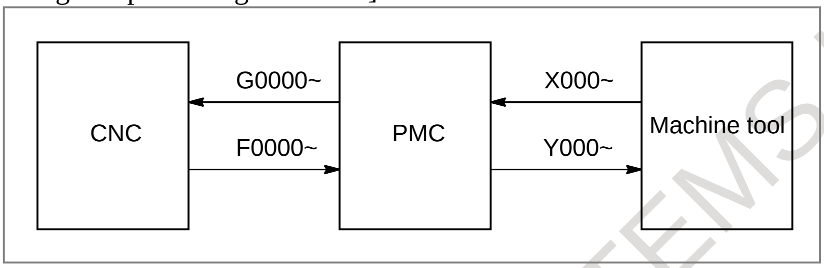
APPENDIX

A INTERFACE BETWEEN CNC AND PMC

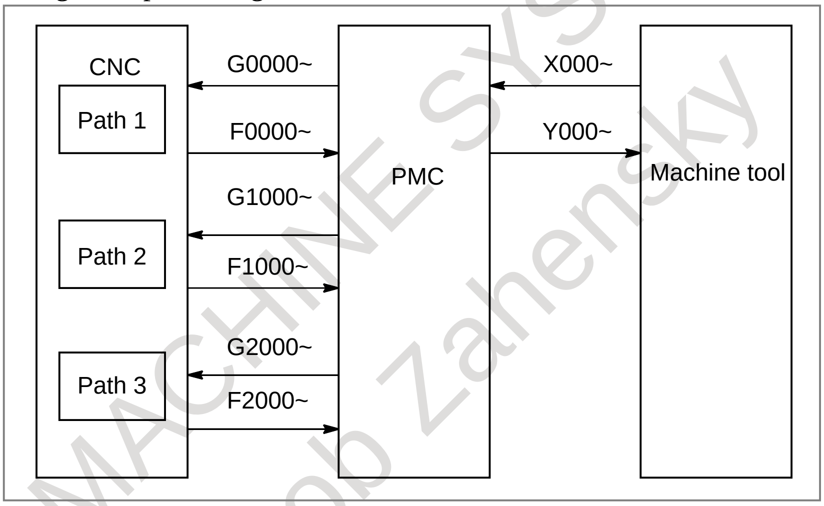
A.1 LIST OF ADDRESSES

Interface addresses among CNC and PMC are as follows:

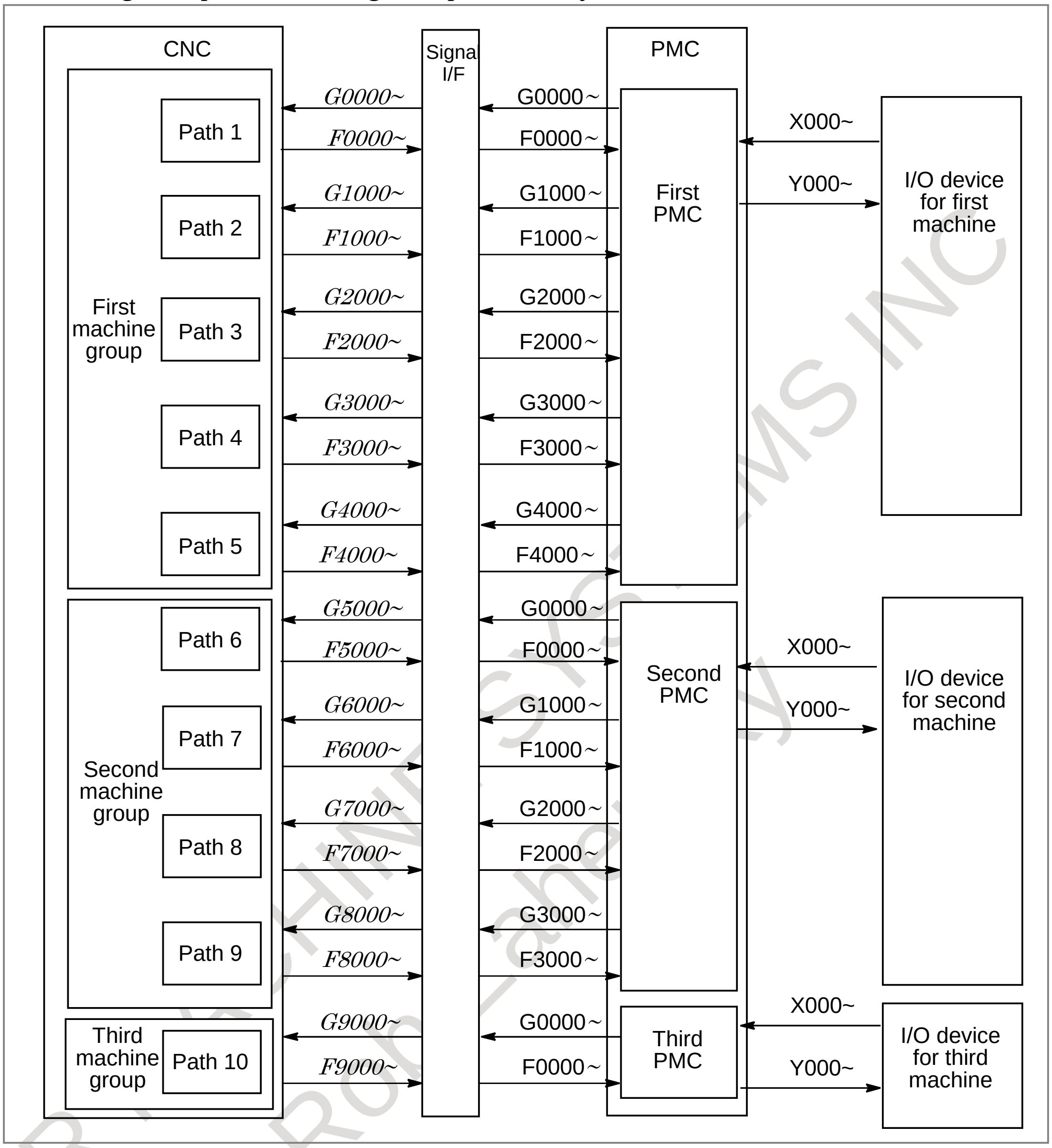
[Example of controlling one path using one PMC]



[Example of controlling three path using one PMC]



[Example of controlling multipath CNC using multipath PMC system]



> **NOTE**
> Each PMC of a multipath PMC system has an independent signal area. The F, G, X, and Y signal addresses of each PMC begin with 0. On the other hand, the F and G signal addresses from the viewpoint of the CNC are fixed for each path number. Note that the F and G signal addresses used in programming of each ladder are different from those from the viewpoint of the CNC.

- Expression of signals

Address	Symbol (#0 to #7 indicates bit position)							
	#7	#6	#5	#4	#3	#2	#1	#0
Fn000	OP	SA	STL	SPL				RWD

In an item where both lathe system and machining center system are described, some signals are covered with shade ( ) in the signal address figure as shown below. This means either lathe system or machining center system does not have this signal. Upper part is for lathe system and lower part is for machining center system.

	#7	#6	#5	#4	#3	#2	#1	#0	
Gn053	*CDZ		ROVLP		UINT			TMRON	T series M series

[Example 1]

The figure above indicates ROVLP is provided only for the lathe system while the other signals for both the lathe system and machining system.

	#7	#6	#5	#4	#3	#2	#1	#0	
Gn040					OFN9	OFN8	OFN7	OFN6	T series M series

[Example 2]

Signals OFN6 to OFN9 are for machining center system only.

NOTE

1 In X addresses in the table, the emergency stop signal for each signal is *ESP<X0008.4>, *ESP<X0008.0>, and *ESP<X008.1>, respectively.

2 For multipath control, one of the following superscripts is attached to the top right of a symbol depending on the signal type.
- Path type (for path 1 on PMC side) : #1
- Path type (for path 2 on PMC side) : #2
- Path type (for path 3 on PMC side) : #3

In addition, #1, #2 or #3 attached to a signal indicates the signal is provided only for path 1, 2, or 3 on PMC side, respectively.
- Path type : #P
- Controlled axis type : #SV
- Spindle type : #SP

In G and F addresses in the table, #P, #SV, or #SP attached to a signal indicates the signal is provided for each path on CNC side, each control axis on CNC side, or each spindle on CNC side, respectively.
- PMC axis control group type: #PX

#PX attached to a signal indicates the signal is provided for each PMC axis control group.

3 For the signals, a single data number is assigned to 8 bits. Each bit has a different meaning.

4 The letter "n" in each address representation indicates the address position used in each path on the CNC side, as shown below.

1st path : n=0 (No. 0 to 999)
2nd path : n=1 (No. 1000 to 1999)
: :
10th path : n=9 (No. 9000 to 9999)

5 For a signal of controlled axis type, when the number of axes eceeds eight for each path, set parameter No. 3021 to address this situation.

6 For a signal of spindle type, when the number of axes eceeds four for each path, set parameter No. 3022 to address this situation.

MT → CNC

Address	Bit number 7	6	5	4	3	2	1	0
X0000								
X0001								
X0002								
X0003								
X0004 T series	SKIP [#1]	ESKIP SKIP6 [#1]	-MIT2[#1] SKIP5 [#1]	+MIT2[#1] SKIP4 [#1]	-MIT1[#1] SKIP3 [#1]	+MIT1[#1] SKIP2 [#1]	SKIP8 [#1]	SKIP7 [#1]
X0004 M series	SKIP [#1]	ESKIP SKIP6 [#1]	SKIP5 [#1]	SKIP4 [#1]	SKIP3 [#1]	SKIP2 [#1]	SKIP8 [#1]	SKIP7 [#1]
X0005								
X0006								
X0007	*DEC8[#2]	*DEC7[#2]	*DEC6[#2]	*DEC5[#2]	*DEC4[#2]	*DEC3[#2]	*DEC2[#2]	*DEC1[#2]
X0008				*ESP			(*ESP)	(*ESP)
X0009	*DEC8[#1]	*DEC7[#1]	*DEC6[#1]	*DEC5[#1]	*DEC4[#1]	*DEC3[#1]	*DEC2[#1]	*DEC1[#1]
X0010	*DEC8[#3]	*DEC7[#3]	*DEC6[#3]	*DEC5[#3]	*DEC4[#3]	*DEC3[#3]	*DEC2[#3]	*DEC1[#3]
X0011 T series	SKIP [#3]	ESKIP[#3] SKIP6 [#3]	-MIT2[#3] SKIP5 [#3]	+MIT2[#3] SKIP4 [#3]	-MIT1[#3] SKIP3 [#3]	+MIT1[#3] SKIP2 [#3]	SKIP8 [#3]	SKIP7 [#3]
X0011 M series	SKIP [#3]	ESKIP[#3] SKIP6 [#3]	SKIP5[#3]	SKIP4[#3]	SKIP3 [#3]	SKIP2 [#3]	SKIP8 [#3]	SKIP7 [#3]
X0012								
X0013 T series	SKIP [#2]	ESKIP[#2] SKIP6 [#2]	-MIT2[#2] SKIP5 [#2]	+MIT2[#2] SKIP4 [#2]	-MIT1[#2] SKIP3 [#2]	+MIT1[#2] SKIP2 [#2]	SKIP8 [#2]	SKIP7 [#2]
X0013 M series	SKIP [#2]	ESKIP[#2] SKIP6 [#2]	SKIP5 [#2]	SKIP4 [#2]	SKIP3 [#2]	SKIP2 [#2]	SKIP8 [#2]	SKIP7 [#2]

PMC → CNC

Address	7	6	5	4	3	2	1	0
Gn000	ED7$^{\#P}$	ED6$^{\#P}$	ED5$^{\#P}$	ED4$^{\#P}$	ED3$^{\#P}$	ED2$^{\#P}$	ED1$^{\#P}$	ED0$^{\#P}$
Gn001	ED15$^{\#P}$	ED14$^{\#P}$	ED13$^{\#P}$	ED12$^{\#P}$	ED11$^{\#P}$	ED10$^{\#P}$	ED9$^{\#P}$	ED8$^{\#P}$
Gn002	ESTB$^{\#P}$	EA6$^{\#P}$	EA5$^{\#P}$	EA4$^{\#P}$	EA3$^{\#P}$	EA2$^{\#P}$	EA1$^{\#P}$	EA0$^{\#P}$
Gn003								
Gn004	MFIN5$^{\#P}$	MFIN4$^{\#P}$	MFIN3$^{\#P}$	MFIN2$^{\#P}$	FIN$^{\#P}$			
Gn005	BFIN$^{\#P}$	AFL$^{\#P}$			TFIN$^{\#P}$	SFIN$^{\#P}$		MFIN$^{\#P}$
Gn006		SKIPP$^{\#P}$		OVC$^{\#P}$		*ABSM$^{\#P}$		SRN$^{\#P}$
Gn007	RLSOT$^{\#P}$	EXLM$^{\#P}$	*FLWU$^{\#P}$	RLSOT3$^{\#P}$		ST$^{\#P}$	STLK$^{\#P}$	RVS$^{\#P}$
Gn008	ERS$^{\#P}$	RRW$^{\#P}$	*SP$^{\#P}$	*ESP$^{\#P}$	*BSL$^{\#P}$		*CSL$^{\#P}$	*IT$^{\#P}$
Gn009				PN16$^{\#P}$	PN8$^{\#P}$	PN4$^{\#P}$	PN2$^{\#P}$	PN1$^{\#P}$
Gn010	*JV7$^{\#P}$	*JV6$^{\#P}$	*JV5$^{\#P}$	*JV4$^{\#P}$	*JV3$^{\#P}$	*JV2$^{\#P}$	*JV1$^{\#P}$	*JV0$^{\#P}$
Gn011	*JV15$^{\#P}$	*JV14$^{\#P}$	*JV13$^{\#P}$	*JV12$^{\#P}$	*JV11$^{\#P}$	*JV10$^{\#P}$	*JV9$^{\#P}$	*JV8$^{\#P}$
Gn012	*FV7$^{\#P}$	*FV6$^{\#P}$	*FV5$^{\#P}$	*FV4$^{\#P}$	*FV3$^{\#P}$	*FV2$^{\#P}$	*FV1$^{\#P}$	*FV0$^{\#P}$
Gn013	*AFV7$^{\#P}$	*AFV6$^{\#P}$	*AFV5$^{\#P}$	*AFV4$^{\#P}$	*AFV3$^{\#P}$	*AFV2$^{\#P}$	*AFV1$^{\#P}$	*AFV0$^{\#P}$
Gn014							ROV2$^{\#P}$	ROV1$^{\#P}$
Gn015								
Gn016	F1D$^{\#P}$							
Gn017								
Gn018	HS2D$^{\#P}$	HS2C$^{\#P}$	HS2B$^{\#P}$	HS2A$^{\#P}$	HS1D$^{\#P}$	HS1C$^{\#P}$	HS1B$^{\#P}$	HS1A$^{\#P}$
Gn019	RT$^{\#P}$	MP4$^{\#P}$	MP2$^{\#P}$	MP1$^{\#P}$	HS3D$^{\#P}$	HS3C$^{\#P}$	HS3B$^{\#P}$	HS3A$^{\#P}$
Gn020					HS4D$^{\#P}$	HS4C$^{\#P}$	HS4B$^{\#P}$	HS4A$^{\#P}$

Address	Bit number 7	6	5	4	3	2	1	0
Gn021	SVR08I#P	SVR07I#P	SVR06I#P	SVR05I#P	SVR04I#P	SVR03I#P	SVR02I#P	SVR01I#P
Gn022	SVSP#P		SVGN#P	DFSYC#P	SVR12I#P	SVR11I#P	SVR10I#P	SVR09I#P
Gn023			NOINPS#P	HREV#P	HNDLF#P			
Gn024	EPN7#P	EPN6#P	EPN5#P	EPN4#P	EPN3#P	EPN2#P	EPN1#P	EPN0#P
Gn025	EPNS#P		EPN13#P	EPN12#P	EPN11#P	EPN10#P	EPN9#P	EPN8#P
Gn026		*SSTP4#SP			SWS4#P		PC4SLC#P	PC3SLC#P
Gn027	CON#P		*SSTP3#SP	*SSTP2#SP	*SSTP1#SP	SWS3#P	SWS2#P	SWS1#P
Gn028	PC2SLC#P	SPSTPA#SP	*SCPFA#SP	*SUCPFA#SP		GR2#SP	GR1#SP	
Gn029		*SSTP#P	SOR#P	SAR#P	GR32#SP	GR31#SP	GR22#SP	GR21#SP
Gn030	SOV7#P	SOV6#P	SOV5#P	SOV4#P	SOV3#P	SOV2#P	SOV1#P	SOV0#P
Gn031	PKESS2#P	PKESS1#P	GR42#SP	GR41#SP	M3R#P			
Gn032	R08I#SP	R07I#SP	R06I#SP	R05I#SP	R04I#SP	R03I#SP	R02I#SP	R01I#SP
Gn033	SIND#SP	SSIN#SP	SGN#SP		R12I#SP	R11I#SP	R10I#SP	R09I#SP
Gn034	R08I2#SP	R07I2#SP	R06I2#SP	R05I2#SP	R04I2#SP	R03I2#SP	R02I2#SP	R01I2#SP
Gn035	SIND2#SP	SSIN2#SP	SGN2#SP		R12I2#SP	R11I2#SP	R10I2#SP	R09I2#SP
Gn036	R08I3#SP	R07I3#SP	R06I3#SP	R05I3#SP	R04I3#SP	R03I3#SP	R02I3#SP	R01I3#SP
Gn037	SIND3#SP	SSIN3#SP	SGN3#SP		R12I3#SP	R11I3#SP	R10I3#SP	R09I3#SP
Gn038	*BECLP#P	*BEUCP#P	SDPC#P		SPPHS#P	SPSYC#P	SBRT#P	
Gn039	GOQSM #P	WOQSM #P	OFN5#P	OFN4#P	OFN3#P	OFN2#P	OFN1#P	OFN0#P
Gn040	WOSET#P	PRC#P	S2TLS#P		OFN9#P	OFN8#P	OFN7#P	OFN6#P
Gn041	HS2ID#P	HS2IC#P	HS2IB#P	HS2IA#P	HS1ID#P	HS1IC#P	HS1IB#P	HS1IA#P
Gn042	DMMC#P				HS3ID#P	HS3IC#P	HS3IB#P	HS3IA#P

Address	Bit 7	6	5	4	3	2	1	0
Gn043	ZRN#P		DNCI#P			MD4#P	MD2#P	MD1#P
Gn044							MLK#P	BDT1#P
Gn045	BDT9#P	BDT8#P	BDT7#P	BDT6#P	BDT5#P	BDT4#P	BDT3#P	BDT2#P
Gn046	DRN#P	KEY4	KEY3	KEY2	KEY1		SBK#P	KEYP
Gn047	TL128#P	TL64#P	TL32#P	TL16#P	TL08#P	TL04#P	TL02#P	TL01#P
Gn048	TLRST#P	TLRSTI#P	TLSKP#P			LFCIV#P	TL512#P	TL256#P
Gn049	*TLV7#P	*TLV6#P	*TLV5#P	*TLV4#P	*TLV3#P	*TLV2#P	*TLV1#P	*TLV0#P
Gn050							*TLV9#P	*TLV8#P
Gn051	*CHLD#P	CHPST#P			*CHP8#P	*CHP4#P	*CHP2#P	*CHP1#P
Gn052								
Gn053	*CDZ#P	SMZ#P	ROVLP#P		UINT#P			TMRON#P
Gn054	UI007#P	UI006#P	UI005#P	UI004#P	UI003#P	UI002#P	UI001#P	UI000#P
Gn055	UI015#P	UI014#P	UI013#P	UI012#P	UI011#P	UI010#P	UI009#P	UI008#P
Gn056	UI023#P	UI022#P	UI021#P	UI020#P	UI019#P	UI018#P	UI017#P	UI016#P
Gn057	UI031#P	UI030#P	UI029#P	UI028#P	UI027#P	UI026#P	UI025#P	UI024#P
Gn058					EXOUT#P	EXSTP#P	EXINP#P	
Gn059	NSYNCA#P						TRRTN#P	TRESC#P
Gn060	*TSB#P							
Gn061	RGTSP4#SP	RGTSP3#SP	RGTSP2#SP	RGTSP1#SP		SYSS#P		RGTAP#P
Gn062	HEAD2	RTNT#P					*CRTOF	
Gn063	NMWT#P	INFD#P	NOZAGC#P		SLSPB#P	SLSPA#P	NOWT	HEAD
Gn064		ESRSYC#P			SLPCB#P	SLPCA#P		

Address	Bit number 7	6	5	4	3	2	1	0
Gn065								
Gn066	EKSET			RTRCT[#P]			ENBKY	IGNVRY[#P]
Gn067	HCREQ	HCABT		EGBS[#P]	MCHK[#P]	MMOD[#P]		MTLC[#P]
Gn068	MTLN07[#P]	MTLN06[#P]	MTLN05[#P]	MTLN04[#P]	MTLN03[#P]	MTLN02[#P]	MTLN01[#P]	MTLN00[#P]
Gn069	MTLN15[#P]	MTLN14[#P]	MTLN13[#P]	MTLN12[#P]	MTLN11[#P]	MTLN10[#P]	MTLN09[#P]	MTLN08[#P]
Gn070	MRDYA[#SP]	ORCMA[#SP]	SFRA[#SP]	SRVA[#SP]	CTH1A[#SP]	CTH2A[#SP]	TLMHA[#SP]	TLMLA[#SP]
Gn071	RCHA[#SP]	RSLA[#SP]	INTGA[#SP]	SOCNA[#SP]	MCFNA[#SP]	SPSLA[#SP]	*ESPA[#SP]	ARSTA[#SP]
Gn072	RCHHGA[#SP]	MFNHGA[#SP]	INCMDA[#SP]	OVRA[#SP]	DEFMDA[#SP]	NRROA[#SP]	ROTAA[#SP]	INDXA[#SP]
Gn073				DSCNA	SORSLA	MPOFA[#SP]	SLVA[#SP]	MORCMA[#SP]
Gn074	MRDYB[#SP]	ORCMB[#SP]	SFRB[#SP]	SRVB[#SP]	CTH1B[#SP]	CTH2B[#SP]	TLMHB[#SP]	TLMLB[#SP]
Gn075	RCHB[#SP]	RSLB[#SP]	INTGB[#SP]	SOCNB[#SP]	MCFNB[#SP]	SPSLB[#SP]	*ESPB[#SP]	ARSTB[#SP]
Gn076	RCHHGB[#SP]	MFNHGB[#SP]	INCMDB[#SP]	OVRB[#SP]	DEFMDB[#SP]	NRROB[#SP]	ROTAB[#SP]	INDXB[#SP]
Gn077				DSCNB	SORSLB	MPOFB[#SP]	SLVB[#SP]	MORCMB[#SP]
Gn078	SH07A[#SP]	SH06A[#SP]	SH05A[#SP]	SH04A[#SP]	SH03A[#SP]	SH02A[#SP]	SH01A[#SP]	SH00A[#SP]
Gn079		SH14A[#SP]	SH13A[#SP]	SH12A[#SP]	SH11A[#SP]	SH10A[#SP]	SH09A[#SP]	SH08A[#SP]
Gn080	SH07B[#SP]	SH06B[#SP]	SH05B[#SP]	SH04B[#SP]	SH03B[#SP]	SH02B[#SP]	SH01B[#SP]	SH00B[#SP]
Gn081		SH14B[#SP]	SH13B[#SP]	SH12B[#SP]	SH11B[#SP]	SH10B[#SP]	SH09B[#SP]	SH08B[#SP]
Gn082	EUI07[#P]	EUI06[#P]	EUI05[#P]	EUI04[#P]	EUI03[#P]	EUI02[#P]	EUI01[#P]	EUI00[#P]
Gn083	EUI15[#P]	EUI14[#P]	EUI13[#P]	EUI12[#P]	EUI11[#P]	EUI10[#P]	EUI09[#P]	EUI08[#P]
Gn084								
Gn085								
Gn086					-Ja[#P]	+Ja[#P]	-Jg[#P]	+Jg[#P]

Address	Bit number 7	6	5	4	3	2	1	0
Gn087	MP42#P	MP41#P		MP32#P	MP31#P		MP22#P	MP21#P
Gn088	HS4ID#P	HS4IC#P	HS4IB#P	HS4IA#P	HNDMP#P			
Gn089								
Gn090	G2SLC#P	G2Y#P	G2Z#P	G2X#P		G2RVY#P	G2RVZ#P	G2RVX#P
Gn091								
Gn092								
Gn093								
Gn094								
Gn095								
Gn096	HROV#P	*HROV6#P	*HROV5#P	*HROV4#P	*HROV3#P	*HROV2#P	*HROV1#P	*HROV0#P
Gn097								
Gn098	EKC7	EKC6	EKC5	EKC4	EKC3	EKC2	EKC1	EKC0
Gn099								
Gn100	+J8#SV	+J7#SV	+J6#SV	+J5#SV	+J4#SV	+J3#SV	+J2#SV	+J1#SV
Gn101	*+ED28#SV	*+ED27#SV	*+ED26#SV	*+ED25#SV	*+ED24#SV	*+ED23#SV	*+ED22#SV	*+ED21#SV
Gn102	-J8#SV	-J7#SV	-J6#SV	-J5#SV	-J4#SV	-J3#SV	-J2#SV	-J1#SV
Gn103	*-ED28#SV	*-ED27#SV	*-ED26#SV	*-ED25#SV	*-ED24#SV	*-ED23#SV	*-ED22#SV	*-ED21#SV
Gn104	+EXL8#SV	+EXL7#SV	+EXL6#SV	+EXL5#SV	+EXL4#SV	+EXL3#SV	+EXL2#SV	+EXL1#SV
Gn105	-EXL8#SV	-EXL7#SV	-EXL6#SV	-EXL5#SV	-EXL4#SV	-EXL3#SV	-EXL2#SV	-EXL1#SV
Gn106	MI8#SV	MI7#SV	MI6#SV	MI5#SV	MI4#SV	MI3#SV	MI2#SV	MI1#SV
Gn107	*+ED38#SV	*+ED37#SV	*+ED36#SV	*+ED35#SV	*+ED34#SV	*+ED33#SV	*+ED32#SV	*+ED31#SV
Gn108	MLK8#SV	MLK7#SV	MLK6#SV	MLK5#SV	MLK4#SV	MLK3#SV	MLK2#SV	MLK1#SV

Address	7	6	5	4	3	2	1	0
Gn109	*-ED38#SV	*-ED37#SV	*-ED36#SV	*-ED35#SV	*-ED34#SV	*-ED33#SV	*-ED32#SV	*-ED31#SV
Gn110	+LM8#SV	+LM7#SV	+LM6#SV	+LM5#SV	+LM4#SV	+LM3#SV	+LM2#SV	+LM1#SV
Gn111								
Gn112	-LM8#SV	-LM7#SV	-LM6#SV	-LM5#SV	-LM4#SV	-LM3#SV	-LM2#SV	-LM1#SV
Gn113								
Gn114	*+L8#SV	*+L7#SV	*+L6#SV	*+L5#SV	*+L4#SV	*+L3#SV	*+L2#SV	*+L1#SV
Gn115								
Gn116	*-L8#SV	*-L7#SV	*-L6#SV	*-L5#SV	*-L4#SV	*-L3#SV	*-L2#SV	*-L1#SV
Gn117								
Gn118	*+ED8#SV	*+ED7#SV	*+ED6#SV	*+ED5#SV	*+ED4#SV	*+ED3#SV	*+ED2#SV	*+ED1#SV
Gn119								
Gn120	*-ED8#SV	*-ED7#SV	*-ED6#SV	*-ED5#SV	*-ED4#SV	*-ED3#SV	*-ED2#SV	*-ED1#SV
Gn121								
Gn122	PK8#SV	PK7#SV	PK6#SV	PK5#SV	PK4#SV	PK3#SV	PK2#SV	PK1#SV
	PKESS2#P	PKESS1#P						
Gn123								
Gn124	DTCH8#SV	DTCH7#SV	DTCH6#SV	DTCH5#SV	DTCH4#SV	DTCH3#SV	DTCH2#SV	DTCH1#SV
Gn125	IUDD8#SV	IUDD7#SV	IUDD6#SV	IUDD5#SV	IUDD4#SV	IUDD3#SV	IUDD2#SV	IUDD1#SV
Gn126	SVF8#SV	SVF7#SV	SVF6#SV	SVF5#SV	SVF4#SV	SVF3#SV	SVF2#SV	SVF1#SV
Gn127								
Gn128	MIX8#SV	MIX7#SV	MIX6#SV	MIX5#SV	MIX4#SV	MIX3#SV	MIX2#SV	MIX1#SV
Gn129								
Gn130	*IT8#SV	*IT7#SV	*IT6#SV	*IT5#SV	*IT4#SV	*IT3#SV	*IT2#SV	*IT1#SV

Address	Bit number 7	6	5	4	3	2	1	0
Gn131								
Gn132	+MIT8#P	+MIT7#P	+MIT6#P	+MIT5#P	+MIT4#P	+MIT3#P	+MIT2#P	+MIT1#P
Gn133								
Gn134	-MIT8#P	-MIT7#P	-MIT6#P	-MIT5#P	-MIT4#P	-MIT3#P	-MIT2#P	-MIT1#P
Gn135								
Gn136	EAX8#SV	EAX7#SV	EAX6#SV	EAX5#SV	EAX4#SV	EAX3#SV	EAX2#SV	EAX1#SV
Gn137								
Gn138	SYNC8#SV	SYNC7#SV	SYNC6#SV	SYNC5#SV	SYNC4#SV	SYNC3#SV	SYNC2#SV	SYNC1#SV
Gn139								
Gn140	SYNCJ8#SV	SYNCJ7#SV	SYNCJ6#SV	SYNCJ5#SV	SYNCJ4#SV	SYNCJ3#SV	SYNCJ2#SV	SYNCJ1#SV
Gn141								
Gn142	EBUFA#PX	ECLRA#PX	ESTPA#PX	ESOFA#PX	ESBKA#PX	EMBUFA#PX	ELCKZA#PX	EFINA#PX
Gn143	EMSBKA#PX	EC6A#PX	EC5A#PX	EC4A#PX	EC3A#PX	EC2A#PX	EC1A#PX	EC0A#PX
Gn144	EIF7A#PX	EIF6A#PX	EIF5A#PX	EIF4A#PX	EIF3A#PX	EIF2A#PX	EIF1A#PX	EIF0A#PX
Gn145	EIF15A#PX	EIF14A#PX	EIF13A#PX	EIF12A#PX	EIF11A#PX	EIF10A#PX	EIF9A#PX	EIF8A#PX
Gn146	EID7A#PX	EID6A#PX	EID5A#PX	EID4A#PX	EID3A#PX	EID2A#PX	EID1A#PX	EID0A#PX
Gn147	EID15A#PX	EID14A#PX	EID13A#PX	EID12A#PX	EID11A#PX	EID10A#PX	EID9A#PX	EID8A#PX
Gn148	EID23A#PX	EID22A#PX	EID21A#PX	EID20A#PX	EID19A#PX	EID18A#PX	EID17A#PX	EID16A#PX
Gn149	EID31A#PX	EID30A#PX	EID29A#PX	EID28A#PX	EID27A#PX	EID26A#PX	EID25A#PX	EID24A#PX
Gn150	EDRN#P	ERT#P	EOVC#P				EROV2#P	EROV1#P
Gn151	*EFOV7#P *EROV7#P	*EFOV6#P *EROV6#P	*EFOV5#P *EROV5#P	*EFOV4#P *EROV4#P	*EFOV3#P *EROV3#P	*EFOV2#P *EROV2#P	*EFOV1#P *EROV1#P	*EFOV0#P *EROV0#P
Gn152								

Address	7	6	5	4	3	2	1	0
Gn153								
Gn154	EBUFB#PX	ECLRB#PX	ESTPB#PX	ESOFB#PX	ESBKB#PX	EMBUFB#PX	ELCKZB#PX	EFINB#PX
Gn155	EMSBKB#PX	EC6B#PX	EC5B#PX	EC4B#PX	EC3B#PX	EC2B#PX	EC1B#PX	EC0B#PX
Gn156	EIF7B#PX	EIF6B#PX	EIF5B#PX	EIF4B#PX	EIF3B#PX	EIF2B#PX	EIF1B#PX	EIF0B#PX
Gn157	EIF15B#PX	EIF14B#PX	EIF13B#PX	EIF12B#PX	EIF11B#PX	EIF10B#PX	EIF9B#PX	EIF8B#PX
Gn158	EID7B#PX	EID6B#PX	EID5B#PX	EID4B#PX	EID3B#PX	EID2B#PX	EID1B#PX	EID0B#PX
Gn159	EID15B#PX	EID14B#PX	EID13B#PX	EID12B#PX	EID11B#PX	EID10B#PX	EID9B#PX	EID8B#PX
Gn160	EID23B#PX	EID22B#PX	EID21B#PX	EID20B#PX	EID19B#PX	EID18B#PX	EID17B#PX	EID16B#PX
Gn161	EID31B#PX	EID30B#PX	EID29B#PX	EID28B#PX	EID27B#PX	EID26B#PX	EID25B#PX	EID24B#PX
Gn162			EOVCB#PX					
Gn163	*EFOV7B#PX *EROV7B#PX	*EFOV6B#PX *EROV6B#PX	*EFOV5B#PX *EROV5B#PX	*EFOV4B#PX *EROV4B#PX	*EFOV3B#PX *EROV3B#PX	*EFOV2B#PX *EROV2B#PX	*EFOV1B#PX *EROV1B#PX	*EFOV0B#PX *EROV0B#PX
Gn164								
Gn165								
Gn166	EBUFC#PX	ECLRC#PX	ESTPC#PX	ESOFC#PX	ESBKC#PX	EMBUFC#PX	ELCKZC#PX	EFINC#PX
Gn167	EMSBKC#PX	EC6C#PX	EC5C#PX	EC4C#PX	EC3C#PX	EC2C#PX	EC1C#PX	EC0C#PX
Gn168	EIF7C#PX	EIF6C#PX	EIF5C#PX	EIF4C#PX	EIF3C#PX	EIF2C#PX	EIF1C#PX	EIF0C#PX
Gn169	EIF15C#PX	EIF14C#PX	EIF13C#PX	EIF12C#PX	EIF11C#PX	EIF10C#PX	EIF9C#PX	EIF8C#PX
Gn170	EID7C#PX	EID6C#PX	EID5C#PX	EID4C#PX	EID3C#PX	EID2C#PX	EID1C#PX	EID0C#PX
Gn171	EID15C#PX	EID14C#PX	EID13C#PX	EID12C#PX	EID11C#PX	EID10C#PX	EID9C#PX	EID8C#PX
Gn172	EID23C#PX	EID22C#PX	EID21C#PX	EID20C#PX	EID19C#PX	EID18C#PX	EID17C#PX	EID16C#PX
Gn173	EID31C#PX	EID30C#PX	EID29C#PX	EID28C#PX	EID27C#PX	EID26C#PX	EID25C#PX	EID24C#PX
Gn174			EOVCC#PX					

Address	Bit number 7	6	5	4	3	2	1	0
Gn175	*EFOV7C[#PX] *EROV7C[#PX]	*EFOV6C[#PX] *EROV6C[#PX]	*EFOV5C[#PX] *EROV5C[#PX]	*EFOV4C[#PX] *EROV4C[#PX]	*EFOV3C[#PX] *EROV3C[#PX]	*EFOV2C[#PX] *EROV2C[#PX]	*EFOV1C[#PX] *EROV1C[#PX]	*EFOV0C[#PX] *EROV0C[#PX]
Gn176								
Gn177								
Gn178	EBUFD[#PX]	ECLRD[#PX]	ESTPD[#PX]	ESOFD[#PX]	ESBKD[#PX]	EMBUFD[#PX]	ELCKZD[#PX]	EFIND[#PX]
Gn179	EMSBKD[#PX]	EC6D[#PX]	EC5D[#PX]	EC4D[#PX]	EC3D[#PX]	EC2D[#PX]	EC1D[#PX]	EC0D[#PX]
Gn180	EIF7D[#PX]	EIF6D[#PX]	EIF5D[#PX]	EIF4D[#PX]	EIF3D[#PX]	EIF2D[#PX]	EIF1D[#PX]	EIF0D[#PX]
Gn181	EIF15D[#PX]	EIF14D[#PX]	EIF13D[#PX]	EIF12D[#PX]	EIF11D[#PX]	EIF10D[#PX]	EIF9D[#PX]	EIF8D[#PX]
Gn182	EID7D[#PX]	EID6D[#PX]	EID5D[#PX]	EID4D[#PX]	EID3D[#PX]	EID2D[#PX]	EID1D[#PX]	EID0D[#PX]
Gn183	EID15D[#PX]	EID14D[#PX]	EID13D[#PX]	EID12D[#PX]	EID11D[#PX]	EID10D[#PX]	EID9D[#PX]	EID8D[#PX]
Gn184	EID23D[#PX]	EID22D[#PX]	EID21D[#PX]	EID20D[#PX]	EID19D[#PX]	EID18D[#PX]	EID17D[#PX]	EID16D[#PX]
Gn185	EID31D[#PX]	EID30D[#PX]	EID29D[#PX]	EID28D[#PX]	EID27D[#PX]	EID26D[#PX]	EID25D[#PX]	EID24D[#PX]
Gn186			EOVCD[#PX]					
Gn187	*EFOV7D[#PX] *EROV7D[#PX]	*EFOV6D[#PX] *EROV6D[#PX]	*EFOV5D[#PX] *EROV5D[#PX]	*EFOV4D[#PX] *EROV4D[#PX]	*EFOV3D[#PX] *EROV3D[#PX]	*EFOV2D[#PX] *EROV2D[#PX]	*EFOV1D[#PX] *EROV1D[#PX]	*EFOV0D[#PX] *EROV0D[#PX]
Gn188								
Gn189								
Gn190	OVLS8[#SV]	OVLS7[#SV]	OVLS6[#SV]	OVLS5[#SV]	OVLS4[#SV]	OVLS3[#SV]	OVLS2[#SV]	OVLS1[#SV]
Gn191								
Gn192	IGVRY8[#SV]	IGVRY7[#SV]	IGVRY6[#SV]	IGVRY5[#SV]	IGVRY4[#SV]	IGVRY3[#SV]	IGVRY2[#SV]	IGVRY1[#SV]
Gn193					HDSR[#P]			
Gn194								
Gn195								
Gn196	*DEC8[#SV]	*DEC7[#SV]	*DEC6[#SV]	*DEC5[#SV]	*DEC4[#SV]	*DEC3[#SV]	*DEC2[#SV]	*DEC1[#SV]

Address	Bit number 7	6	5	4	3	2	1	0
Gn197					MTD$^{\#P}$	MTC$^{\#P}$	MTB$^{\#P}$	MTA$^{\#P}$
Gn198	NPOS8$^{\#SV}$	NPOS7$^{\#SV}$	NPOS6$^{\#SV}$	NPOS5$^{\#SV}$	NPOS4$^{\#SV}$	NPOS3$^{\#SV}$	NPOS2$^{\#SV}$	NPOS1$^{\#SV}$
Gn199							IOLBH2	IOLBH1
Gn200								
Gn201								
Gn202	NDCAL8$^{\#SV}$	NDCAL7$^{\#SV}$	NDCAL6$^{\#SV}$	NDCAL5$^{\#SV}$	NDCAL4$^{\#SV}$	NDCAL3$^{\#SV}$	NDCAL2$^{\#SV}$	NDCAL1$^{\#SV}$
Gn203	PWFL				ESTPR$^{\#P}$			
Gn204	MRDYC$^{\#SP}$	ORCMC$^{\#SP}$	SFRC$^{\#SP}$	SRVC$^{\#SP}$	CTH1C$^{\#SP}$	CTH2C$^{\#SP}$	TLMHC$^{\#SP}$	TLMLC$^{\#SP}$
Gn205	RCHC$^{\#SP}$	RSLC$^{\#SP}$	INTGC$^{\#SP}$	SOCNC$^{\#SP}$	MCFNC$^{\#SP}$	SPSLC$^{\#SP}$	*ESPC$^{\#SP}$	ARSTC$^{\#SP}$
Gn206	RCHHGC$^{\#SP}$	MFNHGC$^{\#SP}$	INCMDC$^{\#SP}$	OVRC$^{\#SP}$	DEFMDC$^{\#SP}$	NRROC$^{\#SP}$	ROTAC$^{\#SP}$	INDXC$^{\#SP}$
Gn207				DSCNC	SORSLC	MPOFC$^{\#SP}$	SLVC$^{\#SP}$	MORCMC$^{\#SP}$
Gn208	SH07C$^{\#SP}$	SH06C$^{\#SP}$	SH05C$^{\#SP}$	SH04C$^{\#SP}$	SH03C$^{\#SP}$	SH02C$^{\#SP}$	SH01C$^{\#SP}$	SH00C$^{\#SP}$
Gn209		SH14C$^{\#SP}$	SH13C$^{\#SP}$	SH12C$^{\#SP}$	SH11C$^{\#SP}$	SH10C$^{\#SP}$	SH09C$^{\#SP}$	SH08C$^{\#SP}$
Gn210	ED23$^{\#P}$	ED22$^{\#P}$	ED21$^{\#P}$	ED20$^{\#P}$	ED19$^{\#P}$	ED18$^{\#P}$	ED17$^{\#P}$	ED16$^{\#P}$
Gn211	ED31$^{\#P}$	ED30$^{\#P}$	ED29$^{\#P}$	ED28$^{\#P}$	ED27$^{\#P}$	ED26$^{\#P}$	ED25$^{\#P}$	ED24$^{\#P}$
Gn212								
Gn213								
Gn214								
Gn215								
Gn216								
Gn217								
Gn220								
Gn251								

Address	Bit 7	6	5	4	3	2	1	0
Gn263								
Gn264					ESSYC4#SP	ESSYC3#SP	ESSYC2#SP	ESSYC1#SP
Gn265					PKESE4#SP	PKESE3#SP	PKESE2#SP	PKESE1#SP
Gn266	MRDYD#SP	ORCMD#SP	SFRD#SP	SRVD#SP	CTH1D#SP	CTH2D#SP	TLMHD#SP	TLMLD#SP
Gn267	RCHD#SP	RSLD#SP	INTGD#SP	SOCND#SP	MCFND#SP	SPSLD#SP	*ESPD#SP	ARSTD#SP
Gn268	RCHHGD#SP	MFNHGD#SP	INCMDD#SP	OVRD#SP	DEFMDD#SP	NRROD#SP	ROTAD#SP	INDXD#SP
Gn269				DSCND	SORSLD	MPOFD#SP	SLVD#SP	MORCMD#SP
Gn270	SH07D#SP	SH06D#SP	SH05D#SP	SH04D#SP	SH03D#SP	SH02D#SP	SH01D#SP	SH00D#SP
Gn271		SH14D#SP	SH13D#SP	SH12D#SP	SH11D#SP	SH10D#SP	SH09D#SP	SH08D#SP
Gn272	R08I4#SP	R07I4#SP	R06I4#SP	R05I4#SP	R04I4#SP	R03I4#SP	R02I4#SP	R01I4#SP
Gn273	SIND4#SP	SSIN4#SP	SGN4#SP		R12I4#SP	R11I4#SP	R10I4#SP	R09I4#SP
Gn274	CSFI4#SP	CSFI3#SP	CSFI2#SP	CSFI1#SP	CONS4#SP	CONS3#SP	CONS2#SP	CONS1#SP
Gn275								
Gn276	UI107#P	UI106#P	UI105#P	UI104#P	UI103#P	UI102#P	UI101#P	UI100#P
Gn277	UI115#P	UI114#P	UI113#P	UI112#P	UI111#P	UI110#P	UI109#P	UI108#P
Gn278	UI123#P	UI122#P	UI121#P	UI120#P	UI119#P	UI118#P	UI117#P	UI116#P
Gn279	UI131#P	UI130#P	UI129#P	UI128#P	UI127#P	UI126#P	UI125#P	UI124#P
Gn280	UI207#P	UI206#P	UI205#P	UI204#P	UI203#P	UI202#P	UI201#P	UI200#P
Gn281	UI215#P	UI214#P	UI213#P	UI212#P	UI211#P	UI210#P	UI209#P	UI208#P
Gn282	UI223#P	UI222#P	UI221#P	UI220#P	UI219#P	UI218#P	UI217#P	UI216#P
Gn283	UI231#P	UI230#P	UI229#P	UI228#P	UI227#P	UI226#P	UI225#P	UI224#P
Gn284	UI307#P	UI306#P	UI305#P	UI304#P	UI303#P	UI302#P	UI301#P	UI300#P

Address	Bit number 7	6	5	4	3	2	1	0
Gn285	UI315$^{\#P}$	UI314$^{\#P}$	UI313$^{\#P}$	UI312$^{\#P}$	UI311$^{\#P}$	UI310$^{\#P}$	UI309$^{\#P}$	UI308$^{\#P}$
Gn286	UI323$^{\#P}$	UI322$^{\#P}$	UI321$^{\#P}$	UI320$^{\#P}$	UI319$^{\#P}$	UI318$^{\#P}$	UI317$^{\#P}$	UI316$^{\#P}$
Gn287	UI331$^{\#P}$	UI330$^{\#P}$	UI329$^{\#P}$	UI328$^{\#P}$	UI327$^{\#P}$	UI326$^{\#P}$	UI325$^{\#P}$	UI324$^{\#P}$
Gn288					SPSYC4$^{\#SP}$	SPSYC3$^{\#SP}$	SPSYC2$^{\#SP}$	SPSYC1$^{\#SP}$
Gn289					SPPHS4$^{\#SP}$	SPPHS3$^{\#SP}$	SPPHS2$^{\#SP}$	SPPHS1$^{\#SP}$
Gn290			PGCK$^{\#P}$					
Gn291								
Gn292								
Gn293								
Gn294								
Gn295	CNCKY	C2SEND						
Gn296	DI8$^{\#SV}$	DI7$^{\#SV}$	DI6$^{\#SV}$	DI5$^{\#SV}$	DI4$^{\#SV}$	DI3$^{\#SV}$	DI2$^{\#SV}$	DI1$^{\#SV}$
Gn297								BCAN$^{\#P}$
Gn298								
Gn299								
Gn300								
Gn301								
Gn302								
Gn303								
Gn304	FRFSMA	INESTRA						
Gn305					HF4A	HF3A	HF2A	HF1A
Gn306						TDFCANA	PWMSEA	

Address	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
Gn307								
Gn308	FRFSMB	INESTRB						
Gn309					HF4B	HF3B	HF2B	HF1B
Gn310						TDFCANB	PWMSEB	
Gn311								
Gn312	FRFSMC	INESTRC						
Gn313					HF4C	HF3C	HF2C	HF1C
Gn314						TDFCANC	PWMSEC	
Gn315								
Gn316	FRFSMD	INESTRD						
Gn317					HF4D	HF3D	HF2D	HF1D
Gn318						TDFCAND	PWMSED	
Gn319								
Gn320								
Gn321								
Gn322								
Gn323								
Gn324								
Gn325								
Gn326								
Gn327								
Gn328	TLRSTI4[#P]	TLRSTI3[#P]	TLRSTI2[#P]	TLRSTI1[#P]	TLRST4[#P]	TLRST3[#P]	TLRST2[#P]	TLRST1[#P]

Address	Bit number 7	6	5	4	3	2	1	0
Gn329	TLNCT4$^{\#P}$	TLNCT3$^{\#P}$	TLNCT2$^{\#P}$	TLNCT1$^{\#P}$	TLSKP4$^{\#P}$	TLSKP3$^{\#P}$	TLSKP2$^{\#P}$	TLSKP1$^{\#P}$
Gn330			TKEY5	TKEY4	TKEY3	TKEY2	TKEY1	TKEY0
Gn331								
Gn332								
Gn333								
Gn334								
Gn335								
Gn336								
Gn337								
Gn338								
Gn339								
Gn340		SLRER$^{\#P}$	SLREF$^{\#P}$					
Gn341	*+ED48$^{\#SV}$	*+ED47$^{\#SV}$	*+ED46$^{\#SV}$	*+ED45$^{\#SV}$	*+ED44$^{\#SV}$	*+ED43$^{\#SV}$	*+ED42$^{\#SV}$	*+ED41$^{\#SV}$
Gn342	*-ED48$^{\#SV}$	*-ED47$^{\#SV}$	*-ED46$^{\#SV}$	*-ED45$^{\#SV}$	*-ED44$^{\#SV}$	*-ED43$^{\#SV}$	*-ED42$^{\#SV}$	*-ED41$^{\#SV}$
Gn343	*+ED58$^{\#SV}$	*+ED57$^{\#SV}$	*+ED56$^{\#SV}$	*+ED55$^{\#SV}$	*+ED54$^{\#SV}$	*+ED53$^{\#SV}$	*+ED52$^{\#SV}$	*+ED51$^{\#SV}$
Gn344	*-ED58$^{\#SV}$	*-ED57$^{\#SV}$	*-ED56$^{\#SV}$	*-ED55$^{\#SV}$	*-ED54$^{\#SV}$	*-ED53$^{\#SV}$	*-ED52$^{\#SV}$	*-ED51$^{\#SV}$
Gn345								
Gn346								
Gn347	NOT3DM$^{\#P}$						HDN$^{\#P}$	
Gn348								
Gn349								
Gn350								

Address	Bit number 7	6	5	4	3	2	1	0
Gn351					SSEGB4#SP	SSEGB3#SP	SSEGB2#SP	SSEGB1#SP
Gn352	*FHRO7#P	*FHRO6#P	*FHRO5#P	*FHRO4#P	*FHRO3#P	*FHRO2#P	*FHRO1#P	*FHRO0#P
Gn353	FHROV#P						*FHRO9#P	*FHRO8#P
Gn354								
Gn355								
Gn356								
Gn357								
Gn358	WPRST8#SV	WPRST7#SV	WPRST6#SV	WPRST5#SV	WPRST4#SV	WPRST3#SV	WPRST2#SV	WPRST1#SV
~								
Gn375								
Gn376	SOV27#P	SOV26#P	SOV25#P	SOV24#P	SOV23#P	SOV22#P	SOV21#P	SOV20#P
Gn377	SOV37#P	SOV36#P	SOV35#P	SOV34#P	SOV33#P	SOV32#P	SOV31#P	SOV30#P
Gn378	SOV47#P	SOV46#P	SOV45#P	SOV44#P	SOV43#P	SOV42#P	SOV41#P	SOV40#P
Gn379	HS5ID#P	HS5IC#P	HS5IB#P	HS5IA#P	HS5D#P	HS5C#P	HS5B#P	HS5A#P
Gn380							MP52#P	MP51#P
Gn381					AUTPHD#P	AUTPHC#P	AUTPHB#P	AUTPHA#P
Gn382								
Gn383								
Gn384								
Gn385								
Gn386								
Gn387								
Gn388								

Address	Bit 7	6	5	4	3	2	1	0
Gn389								
Gn390								
Gn391								
Gn392								
Gn393								
Gn394								
Gn395								
Gn396								
Gn397								
Gn398								
Gn399								
Gn400					*SUCPFD#SP	*SUCPFC#SP	*SUCPFB#SP	
Gn401					*SCPFD#SP	*SCPFC#SP	*SCPFB#SP	
Gn402					SPSTPD#SP	SPSTPC#SP	SPSTPB#SP	
Gn403			SLPCD#P	SLPCC#P			SLSPD#P	SLSPC#P
Gn404								
Gn405								
Gn406	ITF08#P	ITF07#P	ITF06#P	ITF05#P	ITF04#P	ITF03#P	ITF02#P	ITF01#P
Gn407							ITF10#P	ITF09#P
Gn408						HEAD4	HEAD3	STCHK#P
Gn409								
Gn410								

Address	Bit 7	6	5	4	3	2	1	0
Gn411	HS4IE#P	HS3IE#P	HS2IE#P	HS1IE#P	HS4E#P	HS3E#S	HS2E#P	HS1E#P
Gn412				HS5IE#P				HS5E#P
~								
Gn512	MCST8#P	MCST7#P	MCST6#P	MCST5#P	MCST4#P	MCST3#P	MCST2#P	MCST1#P
Gn513	MCST16#P	MCST15#P	MCST14#P	MCST13#P	MCST12#P	MCST11#P	MCST10#P	MCST9#P
Gn514				HPMRSV				MCFIN#P
Gn515								
Gn516								
Gn517	SYPST#P	QRSTD#P				GAE3#P	GAE2#P	GAE1#P
Gn518				DNTCLR				
Gn519								
Gn520								
Gn521	SRVON8#SV	SRVON7#SV	SRVON6#SV	SRVON5#SV	SRVON4#SV	SRVON3#SV	SRVON2#SV	SRVON1#SV
Gn522								
Gn523	SVRVS8#SV	SVRVS7#SV	SVRVS6#SV	SVRVS5#SV	SVRVS4#SV	SVRVS3#SV	SVRVS2#SV	SVRVS1#SV
Gn524								
Gn525	MT8N07#P	MT8N06#P	MT8N05#P	MT8N04#P	MT8N03#P	MT8N02#P	MT8N01#P	MT8N00#P
Gn526	MT8N15#P	MT8N14#P	MT8N13#P	MT8N12#P	MT8N11#P	MT8N10#P	MT8N09#P	MT8N08#P
Gn527	MT8N23#P	MT8N22#P	MT8N21#P	MT8N20#P	MT8N19#P	MT8N18#P	MT8N17#P	MT8N16#P
Gn528	MT8N31#P	MT8N30#P	MT8N29#P	MT8N28#P	MT8N27#P	MT8N26#P	MT8N25#P	MT8N24#P
Gn529								
Gn530	EGBS8#SV	EGBS7#SV	EGBS6#SV	EGBS5#SV	EGBS4#SV	EGBS3#SV	EGBS2#SV	EGBS1#SV
Gn531	EXLM3#P	EXLM2#P		OVLN#P	HBTRN#P		MRVM#P	FWSTP#P
Gn532								

Address	Bit number 7	6	5	4	3	2	1	0
Gn533				SSRS[#P]	SSR4[#SP]	SSR3[#SP]	SSR2[#SP]	SSR1[#SP]
Gn534						AXC4[#P]	AXC2[#P]	AXC1[#P]
Gn535								
Gn536	SPSP[#P]		DASN	EXCST[#P]	ASNST[#P]	RMVST[#P]		
~								
Gn544				MHLC5[#P]	MHLC4[#P]	MHLC3[#P]	MHLC2[#P]	MHLC1[#P]
Gn545				MHUS5[#P]	MHUS4[#P]	MHUS3[#P]	MHUS2[#P]	MHUS1[#P]
Gn546	GQSMC[#P]		OFNC5[#P]	OFNC4[#P]	OFNC3[#P]	OFNC2[#P]	OFNC1[#P]	OFNC0[#P]
Gn547		ONSC[#P]			OFNC9[#P]	OFNC8[#P]	OFNC7[#P]	OFNC6[#P]
Gn548	*CL8[#SV]	*CL7[#SV]	*CL6[#SV]	*CL5[#SV]	*CL4[#SV]	*CL3[#SV]	*CL2[#SV]	*CL1[#SV]
Gn549		ASTC[#P]	RMTC[#P]	GTMSR[#P]				
~								
Gn570								
Gn571								
Gn572								
Gn573								
Gn574	P1SI8[#P]	P1SI7[#P]	P1SI6[#P]	P1SI5[#P]	P1SI4[#P]	P1SI3[#P]	P1SI2[#P]	P1SI1[#P]
Gn575	P1SI16[#P]	P1SI15[#P]	P1SI14[#P]	P1SI13[#P]	P1SI12[#P]	P1SI11[#P]	P1SI10[#P]	P1SI9[#P]
Gn576	P2SI8[#P]	P2SI7[#P]	P2SI6[#P]	P2SI5[#P]	P2SI4[#P]	P2SI3[#P]	P2SI2[#P]	P2SI1[#P]
Gn577	P2SI16[#P]	P2SI15[#P]	P2SI14[#P]	P2SI13[#P]	P2SI12[#P]	P2SI11[#P]	P2SI10[#P]	P2SI9[#P]
Gn578	PSIM[#P]						PSI2[#P]	PSI1[#P]
Gn579		NHSW	WBEND					
Gn580	*ACTF8[#SV]	*ACTF7[#SV]	*ACTF6[#SV]	*ACTF5[#SV]	*ACTF4[#SV]	*ACTF3[#SV]	*ACTF2[#SV]	*ACTF1[#SV]
Gn581	SLANG	LANG7	LANG6	LANG5	LANG4	LANG3	LANG2	LANG1
~								

Gn586	MDOFF4#SP	MDOFF3#SP	MDOFF2#SP	MDOFF1#SP				
Gn587	SPAPH4	SPAPH3	SPAPH2	SPAPH1	SPMST4	SPMST3	SPMST2	SPMST1
Gn588	SMSL24	SMSL23	SMSL22	SMSL21	SMSL14	SMSL13	SMSL12	SMSL11
Gn589								
~								
Gn592								
Gn594	OTD7#P	OTD6#P	OTD5#P	OTD4#P	OTD3#P	OTD2#P	OTD1#P	OTD0#P
Gn595	OTD15#P	OTD14#P	OTD13#P	OTD12#P	OTD11#P	OTD10#P	OTD9#P	OTD8#P
Gn596	OTA8	OTA7	OTA6	OTA5	OTA4	OTA3	OTA2	OTA1
Gn597	-OT3#P	+OT3#P	-OT2#P	+OT2#P	-OT12#P	+OT12#P	-OT11#P	+OT11#P
Gn598	-OT3C#P	+OT3C#P	-OT2C#P	+OT2C#P	-OT12C#P	+OT12C#P	-OT11C#P	+OT11C#P
Gn599	RTC3#P	RTC2#P	CTC3#P	CTC2#P	GIS#P			OTSW#P
~								
Gn687	SVMWC8#SV	SVMWC7#SV	SVMWC6#SV	SVMWC5#SV	SVMWC4#SV	SVMWC3#SV	SVMWC2#SV	SVMWC1#SV
~								
Gn708	RE08I#SP	RE07I#SP	RE06I#SP	RE05I#SP	RE04I#SP	RE03I#SP	RE02I#SP	RE01I#SP
Gn709	RE16I#SP	RE15I#SP	RE14I#SP	RE13I#SP	RE12I#SP	RE11I#SP	RE10I#SP	RE09I#SP
Gn710	RE24I#SP	RE23I#SP	RE22I#SP	RE21I#SP	RE20I#SP	RE19I#SP	RE18I#SP	RE17I#SP
Gn711	RE32I#SP	RE31I#SP	RE30I#SP	RE29I#SP	RE28I#SP	RE27I#SP	RE26I#SP	RE25I#SP
Gn712	RE08I2#SP	RE07I2#SP	RE06I2#SP	RE05I2#SP	RE04I2#SP	RE03I2#SP	RE02I2#SP	RE01I2#SP
Gn713	RE16I2#SP	RE15I2#SP	RE14I2#SP	RE13I2#SP	RE12I2#SP	RE11I2#SP	RE10I2#SP	RE09I2#SP
Gn714	RE24I2#SP	RE23I2#SP	RE22I2#SP	RE21I2#SP	RE20I2#SP	RE19I2#SP	RE18I2#SP	RE17I2#SP
Gn715	RE32I2#SP	RE31I2#SP	RE30I2#SP	RE29I2#SP	RE28I2#SP	RE27I2#SP	RE26I2#SP	RE25I2#SP
Gn716	RE08I3#SP	RE07I3#SP	RE06I3#SP	RE05I3#SP	RE04I3#SP	RE03I3#SP	RE02I3#SP	RE01I3#SP
Gn717	RE16I3#SP	RE15I3#SP	RE14I3#SP	RE13I3#SP	RE12I3#SP	RE11I3#SP	RE10I3#SP	RE09I3#SP
Gn718	RE24I3#SP	RE23I3#SP	RE22I3#SP	RE21I3#SP	RE20I3#SP	RE19I3#SP	RE18I3#SP	RE17I3#SP

Gn719	RE32I3$^{\#SP}$	RE31I3$^{\#SP}$	RE30I3$^{\#SP}$	RE29I3$^{\#SP}$	RE28I3$^{\#SP}$	RE27I3$^{\#SP}$	RE26I3$^{\#SP}$	RE25I3$^{\#SP}$
Gn720	RE08I4$^{\#SP}$	RE07I4$^{\#SP}$	RE06I4$^{\#SP}$	RE05I4$^{\#SP}$	RE04I4$^{\#SP}$	RE03I4$^{\#SP}$	RE02I4$^{\#SP}$	RE01I4$^{\#SP}$
Gn721	RE16I4$^{\#SP}$	RE15I4$^{\#SP}$	RE14I4$^{\#SP}$	RE13I4$^{\#SP}$	RE12I4$^{\#SP}$	RE11I4$^{\#SP}$	RE10I4$^{\#SP}$	RE09I4$^{\#SP}$
Gn722	RE24I4$^{\#SP}$	RE23I4$^{\#SP}$	RE22I4$^{\#SP}$	RE21I4$^{\#SP}$	RE20I4$^{\#SP}$	RE19I4$^{\#SP}$	RE18I4$^{\#SP}$	RE17I4$^{\#SP}$
Gn723	RE32I4$^{\#SP}$	RE31I4$^{\#SP}$	RE30I4$^{\#SP}$	RE29I4$^{\#SP}$	RE28I4$^{\#SP}$	RE27I4$^{\#SP}$	RE26I4$^{\#SP}$	RE25I4$^{\#SP}$
Gn724								
~								
Gn726	TDC8$^{\#SV}$	TDC7$^{\#SV}$	TDC6$^{\#SV}$	TDC5$^{\#SV}$	TDC4$^{\#SV}$	TDC3$^{\#SV}$	TDC2$^{\#SV}$	TDC1$^{\#SV}$
~								
Gn765	TPMG07	TPMG06	TPMG05	TPMG04	TPMG03	TPMG02	TPMG01	TPMG00
Gn766								
Gn767								

CNC → PMC

Address	7	6	5	4	3	2	1	0
Fn000	OP$^{\#P}$	SA$^{\#P}$	STL$^{\#P}$	SPL$^{\#P}$				RWD$^{\#P}$
Fn001	MA$^{\#P}$		TAP$^{\#P}$	ENB$^{\#SP}$	DEN$^{\#P}$	BAL$^{\#P}$	RST$^{\#P}$	AL$^{\#P}$
Fn002	MDRN$^{\#P}$	CUT$^{\#P}$		SRNMV$^{\#P}$	THRD$^{\#P}$	CSS$^{\#P}$	RPDO$^{\#P}$	INCH$^{\#P}$
Fn003		MEDT$^{\#P}$	MMEM$^{\#P}$	MRMT$^{\#P}$	MMDI$^{\#P}$	MJ$^{\#P}$	MH$^{\#P}$	MINC$^{\#P}$
Fn004			MREF$^{\#P}$	MAFL$^{\#P}$	MSBK$^{\#P}$	MABSM$^{\#P}$	MMLK$^{\#P}$	MBDT1$^{\#P}$
Fn005	MBDT9$^{\#P}$	MBDT8$^{\#P}$	MBDT7$^{\#P}$	MBDT6$^{\#P}$	MBDT5$^{\#P}$	MBDT4$^{\#P}$	MBDT3$^{\#P}$	MBDT2$^{\#P}$
Fn006						ERTVA$^{\#P}$	MDIRST$^{\#P}$	TPPRS
Fn007	BF$^{\#P}$				TF$^{\#P}$	SF$^{\#P}$		MF$^{\#P}$
Fn008	MF5$^{\#P}$	MF4$^{\#P}$	MF3$^{\#P}$	MF2$^{\#P}$				
Fn009	DM00$^{\#P}$	DM01$^{\#P}$	DM02$^{\#P}$	DM30$^{\#P}$				
Fn010	M07$^{\#P}$	M06$^{\#P}$	M05$^{\#P}$	M04$^{\#P}$	M03$^{\#P}$	M02$^{\#P}$	M01$^{\#P}$	M00$^{\#P}$
Fn011	M15$^{\#P}$	M14$^{\#P}$	M13$^{\#P}$	M12$^{\#P}$	M11$^{\#P}$	M10$^{\#P}$	M09$^{\#P}$	M08$^{\#P}$
Fn012	M23$^{\#P}$	M22$^{\#P}$	M21$^{\#P}$	M20$^{\#P}$	M19$^{\#P}$	M18$^{\#P}$	M17$^{\#P}$	M16$^{\#P}$
Fn013	M31$^{\#P}$	M30$^{\#P}$	M29$^{\#P}$	M28$^{\#P}$	M27$^{\#P}$	M26$^{\#P}$	M25$^{\#P}$	M24$^{\#P}$
Fn014	M207$^{\#P}$	M206$^{\#P}$	M205$^{\#P}$	M204$^{\#P}$	M203$^{\#P}$	M202$^{\#P}$	M201$^{\#P}$	M200$^{\#P}$
Fn015	M215$^{\#P}$	M214$^{\#P}$	M213$^{\#P}$	M212$^{\#P}$	M211$^{\#P}$	M210$^{\#P}$	M209$^{\#P}$	M208$^{\#P}$
Fn016	M307$^{\#P}$	M306$^{\#P}$	M305$^{\#P}$	M304$^{\#P}$	M303$^{\#P}$	M302$^{\#P}$	M301$^{\#P}$	M300$^{\#P}$
	M223$^{\#P}$	M222$^{\#P}$	M221$^{\#P}$	M220$^{\#P}$	M219$^{\#P}$	M218$^{\#P}$	M217$^{\#P}$	M216$^{\#P}$
Fn017	M315$^{\#P}$	M314$^{\#P}$	M313$^{\#P}$	M312$^{\#P}$	M311$^{\#P}$	M310$^{\#P}$	M309$^{\#P}$	M308$^{\#P}$
	M231$^{\#P}$	M230$^{\#P}$	M229$^{\#P}$	M228$^{\#P}$	M227$^{\#P}$	M226$^{\#P}$	M225$^{\#P}$	M224$^{\#P}$
Fn018								
Fn019								
Fn020								

Address	7	6	5	4	3	2	1	0
Fn021								
Fn022	S07#P	S06#P	S05#P	S04#P	S03#P	S02#P	S01#P	S00#P
Fn023	S15#P	S14#P	S13#P	S12#P	S11#P	S10#P	S09#P	S08#P
Fn024	S23#P	S22#P	S21#P	S20#P	S19#P	S18#P	S17#P	S16#P
Fn025	S31#P	S30#P	S29#P	S28#P	S27#P	S26#P	S25#P	S24#P
Fn026	T07#P	T06#P	T05#P	T04#P	T03#P	T02#P	T01#P	T00#P
Fn027	T15#P	T14#P	T13#P	T12#P	T11#P	T10#P	T09#P	T08#P
Fn028	T23#P	T22#P	T21#P	T20#P	T19#P	T18#P	T17#P	T16#P
Fn029	T31#P	T30#P	T29#P	T28#P	T27#P	T26#P	T25#P	T24#P
Fn030	B07#P	B06#P	B05#P	B04#P	B03#P	B02#P	B01#P	B00#P
Fn031	B15#P	B14#P	B13#P	B12#P	B11#P	B10#P	B09#P	B08#P
Fn032	B23#P	B22#P	B21#P	B20#P	B19#P	B18#P	B17#P	B16#P
Fn033	B31#P	B30#P	B29#P	B28#P	B27#P	B26#P	B25#P	B24#P
Fn034	SRSRDY#P	SRSP1R#SP	SRSP2R#SP	SRSP3R#SP	SRSP4R#SP	GR3O#P	GR2O#P	GR1O#P
Fn035								SPAL#P
Fn036	R08O#SP	R07O#SP	R06O#SP	R05O#SP	R04O#SP	R03O#SP	R02O#SP	R01O#SP
Fn037					R12O#SP	R11O#SP	R10O#SP	R09O#SP
Fn038					ENB3#SP	ENB2#SP	SUCLPA#SP	SCLPA#SP
Fn039					CHPCYL#P	CHPMD#P	ENB4#SP	MSPOSA#SP
Fn040	AR07#SP	AR06#SP	AR05#SP	AR04#SP	AR03#SP	AR02#SP	AR01#SP	AR00#SP
Fn041	AR15#SP	AR14#SP	AR13#SP	AR12#SP	AR11#SP	AR10#SP	AR09#SP	AR08#SP
Fn042								

Address	7	6	5	4	3	2	1	0
Fn043					SYCAL4#SP	SYCAL3#SP	SYCAL2#SP	SYCAL1#SP
Fn044				SYCAL#P	FSPPH#P	FSPSY#P	FSCSL#P	
Fn045	ORARA#SP	TLMA#SP	LDT2A#SP	LDT1A#SP	SARA#SP	SDTA#SP	SSTA#SP	ALMA#SP
Fn046	MORA2A#SP	MORA1A#SP	PORA2A#SP	SLVSA#SP	RCFNA#SP	RCHPA#SP	CFINA#SP	CHPA#SP
Fn047				EXOFA	SORENA		INCSTA#SP	PC1DTA#SP
Fn048				CSPENA#SP				
Fn049	ORARB#SP	TLMB#SP	LDT2B#SP	LDT1B#SP	SARB#SP	SDTB#SP	SSTB#SP	ALMB#SP
Fn050	MORA2B#SP	MORA1B#SP	PORA2B#SP	SLVSB#SP	RCFNB#SP	RCHPB#SP	CFINB#SP	CHPB#SP
Fn051				EXOFB	SORENB		INCSTB#SP	PC1DTB#SP
Fn052				CSPENB#SP				
Fn053	EKENB			BGEACT#P	IOALM#P	IOBSY#P	PRGDPL	INHKY
Fn054	UO007#P	UO006#P	UO005#P	UO004#P	UO003#P	UO002#P	UO001#P	UO000#P
Fn055	UO015#P	UO014#P	UO013#P	UO012#P	UO011#P	UO010#P	UO009#P	UO008#P
Fn056	UO107#P	UO106#P	UO105#P	UO104#P	UO103#P	UO102#P	UO101#P	UO100#P
Fn057	UO115#P	UO114#P	UO113#P	UO112#P	UO111#P	UO110#P	UO109#P	UO108#P
Fn058	UO123#P	UO122#P	UO121#P	UO120#P	UO119#P	UO118#P	UO117#P	UO116#P
Fn059	UO131#P	UO130#P	UO129#P	UO128#P	UO127#P	UO126#P	UO125#P	UO124#P
Fn060						ESCAN#P	ESEND#P	EREND#P
Fn061			MTLA#P	MTLANG#P	HCEXE	HCAB2	BCLP#P	BUCLP#P
Fn062	PRTSF#P	D3ROT#P		S2MES#P	S1MES#P			AICC#P
Fn063		WATO#P		COSP2#P	COSP1#P			
Fn064	TIALM#P	TICHK#P	COSP#P		TLCHB#P	TLCHI#P	TLNW#P	TLCH#P

Address	Bit number 7	6	5	4	3	2	1	0
Fn065		SYNMOD#P		RTRCTF#P		RSMAX#P	RGSPM#P	RGSPP#P
Fn066			PECK2#P			FEED0#P	RTPT#P	
Fn067								
Fn068								
Fn069								
Fn070	PSW08#P	PSW07#P	PSW06#P	PSW05#P	PSW04#P	PSW03#P	PSW02#P	PSW01#P
Fn071	PSW16#P	PSW15#P	PSW14#P	PSW13#P	PSW12#P	PSW11#P	PSW10#P	PSW09#P
Fn072	OUT7#P	OUT6#P	OUT5#P	OUT4#P	OUT3#P	OUT2#P	OUT1#P	OUT0#P
Fn073				ZRNO#P		MD4O#P	MD2O#P	MD1O#P
Fn074	OUT15#P	OUT14#P	OUT13#P	OUT12#P	OUT11#P	OUT10#P	OUT9#P	OUT8#P
Fn075	SPO#P	KEYO	DRNO#P	MLKO#P	SBKO#P	BDTO#P		
Fn076			ROV2O#P	ROV1O#P	RTAP#P		MP2O#P	MP1O#P
Fn077		RTO#P			HS1DO#P	HS1CO#P	HS1BO#P	HS1AO#P
Fn078	*FV7O#P	*FV6O#P	*FV5O#P	*FV4O#P	*FV3O#P	*FV2O#P	*FV1O#P	*FV0O#P
Fn079	*JV7O#P	*JV6O#P	*JV5O#P	*JV4O#P	*JV3O#P	*JV2O#P	*JV1O#P	*JV0O#P
Fn080	*JV15O#P	*JV14O#P	*JV13O#P	*JV12O#P	*JV11O#P	*JV10O#P	*JV9O#P	*JV8O#P
Fn081	-J4O#P	+J4O#P	-J3O#P	+J3O#P	-J2O#P	+J2O#P	-J1O#P	+J1O#P
Fn082		EGBSM#P				RVSL#P		
Fn083								
Fn084	EUO07#P	EUO06#P	EUO05#P	EUO04#P	EUO03#P	EUO02#P	EUO01#P	EUO00#P
Fn085	EUO15#P	EUO14#P	EUO13#P	EUO12#P	EUO11#P	EUO10#P	EUO09#P	EUO08#P
Fn086								

Address	Bit number 7	6	5	4	3	2	1	0
Fn087								
Fn088								
Fn089								
Fn090	SVSPM#P	SVAR#P	SYSSM#P	SYAR#P	ABTSP3#SP	ABTSP2#SP	ABTSP1#SP	ABTQSV#P
Fn091			ADCO#P	ABTSP4#SP	MMMOD#P	MRVSP#P	MNCHG#P	MRVMD#P
Fn092			TRSPS#P	TRMTN#P	TRACT#P			
Fn093	SVWRN4#P	SVWRN3#P	SVWRN2#P	SVWRN1#P	WFAN#P	LFCIF#P	SFAN#P	LIFOVR#P
Fn094	ZP8#SV	ZP7#SV	ZP6#SV	ZP5#SV	ZP4#SV	ZP3#SV	ZP2#SV	ZP1#SV
Fn095								
Fn096	ZP28#SV	ZP27#SV	ZP26#SV	ZP25#SV	ZP24#SV	ZP23#SV	ZP22#SV	ZP21#SV
Fn097								
Fn098	ZP38#SV	ZP37#SV	ZP36#SV	ZP35#SV	ZP34#SV	ZP33#SV	ZP32#SV	ZP31#SV
Fn099								
Fn100	ZP48#SV	ZP47#SV	ZP46#SV	ZP45#SV	ZP44#SV	ZP43#SV	ZP42#SV	ZP41#SV
Fn101								
Fn102	MV8#SV	MV7#SV	MV6#SV	MV5#SV	MV4#SV	MV3#SV	MV2#SV	MV1#SV
Fn103								
Fn104	INP8#SV	INP7#SV	INP6#SV	INP5#SV	INP4#SV	INP3#SV	INP2#SV	INP1#SV
Fn105								
Fn106	MVD8#SV	MVD7#SV	MVD6#SV	MVD5#SV	MVD4#SV	MVD3#SV	MVD2#SV	MVD1#SV
Fn107								
Fn108	MMI8#SV	MMI7#SV	MMI6#SV	MMI5#SV	MMI4#SV	MMI3#SV	MMI2#SV	MMI1#SV

Address	Bit number 7	6	5	4	3	2	1	0
Fn109								
Fn110	MDTCH8#SV	MDTCH7#SV	MDTCH6#SV	MDTCH5#SV	MDTCH4#SV	MDTCH3#SV	MDTCH2#SV	MDTCH1#SV
Fn111								
Fn112	EADEN8#SV	EADEN7#SV	EADEN6#SV	EADEN5#SV	EADEN4#SV	EADEN3#SV	EADEN2#SV	EADEN1#SV
Fn113								
Fn114	TRQL8#SV	TRQL7#SV	TRQL6#SV	TRQL5#SV	TRQL4#SV	TRQL3#SV	TRQL2#SV	TRQL1#SV
Fn115								
Fn116								
Fn117								
Fn118	SYN8O#SV	SYN7O#SV	SYN6O#SV	SYN5O#SV	SYN4O#SV	SYN3O#SV	SYN2O#SV	SYN1O#SV
Fn119								
Fn120	ZRF8#SV	ZRF7#SV	ZRF6#SV	ZRF5#SV	ZRF4#SV	ZRF3#SV	ZRF2#SV	ZRF1#SV
Fn121								
Fn122	HDO7#P	HDO6#P	HDO5#P	HDO4#P	HDO3#P	HDO2#P	HDO1#P	HDO0#P
Fn123								
Fn124	+OT8#SV	+OT7#SV	+OT6#SV	+OT5#SV	+OT4#SV	+OT3#SV	+OT2#SV	+OT1#SV
Fn125								
Fn126	-OT8#SV	-OT7#SV	-OT6#SV	-OT5#SV	-OT4#SV	-OT3#SV	-OT2#SV	-OT1#SV
Fn127								
Fn128								
Fn129	*EAXSL#P		EOV0#P					
Fn130	EBSYA#PX	EOTNA#PX	EOTPA#PX	EGENA#PX	EDENA#PX	EIALA#PX	ECKZA#PX	EINPA#PX

Address	Bit number 7	6	5	4	3	2	1	0
Fn131					EMF3A#PX	EMF2A#PX	EABUFA#PX	EMFA#PX
Fn132	EM28A#PX	EM24A#PX	EM22A#PX	EM21A#PX	EM18A#PX	EM14A#PX	EM12A#PX	EM11A#PX
Fn133	EBSYB#PX	EOTNB#PX	EOTPB#PX	EGENB#PX	EDENB#PX	EIALB#PX	ECKZB#PX	EINPB#PX
Fn134					EMF3B#PX	EMF2B#PX	EABUFB#PX	EMFB#PX
Fn135	EM28B#PX	EM24B#PX	EM22B#PX	EM21B#PX	EM18B#PX	EM14B#PX	EM12B#PX	EM11B#PX
Fn136	EBSYC#PX	EOTNC#PX	EOTPC#PX	EGENC#PX	EDENC#PX	EIALC#PX	ECKZC#PX	EINPC#PX
Fn137					EMF3C#PX	EMF2C#PX	EABUFC#PX	EMFC#PX
Fn138	EM28C#PX	EM24C#PX	EM22C#PX	EM21C#PX	EM18C#PX	EM14C#PX	EM12C#PX	EM11C#PX
Fn139	EBSYD#PX	EOTND#PX	EOTPD#PX	EGEND#PX	EDEND#PX	EIALD#PX	ECKZD#PX	EINPD#PX
Fn140					EMF3D#PX	EMF2D#PX	EABUFD#PX	EMFD#PX
Fn141	EM28D#PX	EM24D#PX	EM22D#PX	EM21D#PX	EM18D#PX	EM14D#PX	EM12D#PX	EM11D#PX
Fn142	EM48A#PX	EM44A#PX	EM42A#PX	EM41A#PX	EM38A#PX	EM34A#PX	EM32A#PX	EM31A#PX
Fn143								
Fn144								
Fn145	EM48B#PX	EM44B#PX	EM42B#PX	EM41B#PX	EM38B#PX	EM34B#PX	EM32B#PX	EM31B#PX
Fn146								
Fn147								
Fn148	EM48C#PX	EM44C#PX	EM42C#PX	EM41C#PX	EM38C#PX	EM34C#PX	EM32C#PX	EM31C#PX
Fn149								
Fn150								
Fn151	EM48D#PX	EM44D#PX	EM42D#PX	EM41D#PX	EM38D#PX	EM34D#PX	EM32D#PX	EM31D#PX
Fn152								

Address	Bit number 7	6	5	4	3	2	1	0
Fn153								
Fn154								TLAL#P
Fn155								
Fn156								
Fn157								
Fn158								
Fn159								
Fn160	MSP07#P	MSP06#P	MSP05#P	MSP04#P	MSP03#P	MSP02#P	MSP01#P	MSP00#P
Fn161	MSP15#P	MSP14#P	MSP13#P	MSP12#P	MSP11#P	MSP10#P	MSP09#P	MSP08#P
Fn162								
Fn163								
Fn164								
Fn165								
Fn166								
Fn167								
Fn168	ORARC#SP	TLMC#SP	LDT2C#SP	LDT1C#SP	SARC#SP	SDTC#SP	SSTC#SP	ALMC#SP
Fn169	MORA2C#SP	MORA1C#SP	PORA2C#SP	SLVSC#SP	RCFNC#SP	RCHPC#SP	CFINC#SP	CHPC#SP
Fn170				EXOFC	SORENC		INCSTC#SP	PC1DTC#SP
Fn171				CSPENC#SP				
Fn172	PBATL#P	PBATZ#P						
Fn173								
Fn174								

Address	7	6	5	4	3	2	1	0
Fn175								
Fn176								
Fn177								
Fn178								
Fn179								
Fn180	CLRCH8#SV	CLRCH7#SV	CLRCH6#SV	CLRCH5#SV	CLRCH4#SV	CLRCH3#SV	CLRCH2#SV	CLRCH1#SV
Fn181								
Fn182	EACNT8#SV	EACNT7#SV	EACNT6#SV	EACNT5#SV	EACNT4#SV	EACNT3#SV	EACNT2#SV	EACNT1#SV
Fn183								
Fn184	ABDT8#SV	ABDT7#SV	ABDT6#SV	ABDT5#SV	ABDT4#SV	ABDT3#SV	ABDT2#SV	ABDT1#SV
Fn185								
Fn186								
Fn187								
Fn188								
Fn189								
Fn190	TRQM8#SV	TRQM7#SV	TRQM6#SV	TRQM5#SV	TRQM4#SV	TRQM3#SV	TRQM2#SV	TRQM1#SV
Fn191								
Fn192								
Fn193								
Fn194								
Fn195								
Fn196								

Address	Bit number 7	6	5	4	3	2	1	0
Fn197					MFSYND#P	MFSYNC#P	MFSYNB#P	MFSYNA#P
Fn198								
Fn199								
Fn200	R08O2#SP	R07O2#SP	R06O2#SP	R05O2#SP	R04O2#SP	R03O2#SP	R02O2#SP	R01O2#SP
Fn201					R12O2#SP	R11O2#SP	R10O2#SP	R09O2#SP
Fn202	AR072#SP	AR062#SP	AR052#SP	AR042#SP	AR032#SP	AR022#SP	AR012#SP	AR002#SP
Fn203	AR152#SP	AR142#SP	AR132#SP	AR122#SP	AR112#SP	AR102#SP	AR092#SP	AR082#SP
Fn204	R08O3#SP	R07O3#SP	R06O3#SP	R05O3#SP	R04O3#SP	R03O3#SP	R02O3#SP	R01O3#SP
Fn205					R12O3#SP	R11O3#SP	R10O3#SP	R09O3#SP
Fn206	AR073#SP	AR063#SP	AR053#SP	AR043#SP	AR033#SP	AR023#SP	AR013#SP	AR003#SP
Fn207	AR153#SP	AR143#SP	AR133#SP	AR123#SP	AR113#SP	AR103#SP	AR093#SP	AR083#SP
Fn208	EGBM8#SV	EGBM7#SV	EGBM6#SV	EGBM5#SV	EGBM4#SV	EGBM3#SV	EGBM2#SV	EGBM1#SV
Fn209								
Fn210	SYNMT8#P	SYNMT7#P	SYNMT6#P	SYNMT5#P	SYNMT4#P	SYNMT3#P	SYNMT2#P	SYNMT1#P
Fn211	SYNOF8#P	SYNOF7#P	SYNOF6#P	SYNOF5#P	SYNOF4#P	SYNOF3#P	SYNOF2#P	SYNOF1#P
Fn212								
Fn213								
Fn214								
Fn215								
Fn216								
Fn217								
Fn218								

Address	Bit number 7	6	5	4	3	2	1	0
Fn263								
Fn264	SPWRN8#P	SPWRN7#P	SPWRN6#P	SPWRN5#P	SPWRN4#P	SPWRN3#P	SPWRN2#P	SPWRN1#P
Fn265								SPWRN9#P
Fn266	ORARD#SP	TLMD#SP	LDT2D#SP	LDT1D#SP	SARD#SP	SDTD#SP	SSTD#SP	ALMD#SP
Fn267	MORA2D#SP	MORA1D#SP	PORA2D#SP	SLVSD#SP	RCFND#SP	RCHPD#SP	CFIND#SP	CHPD#SP
Fn268				EXOFD	SOREND		INCSTD#SP	PC1DTD#SP
Fn269				CSPEND#SP				
Fn270	R08O4#SP	R07O4#SP	R06O4#SP	R05O4#SP	R04O4#SP	R03O4#SP	R02O4#SP	R01O4#SP
Fn271					R12O4#SP	R11O4#SP	R10O4#SP	R09O4#SP
Fn272	AR074#SP	AR064#SP	AR054#SP	AR044#SP	AR034#SP	AR024#SP	AR014#SP	AR004#SP
Fn273	AR154#SP	AR144#SP	AR134#SP	AR124#SP	AR114#SP	AR104#SP	AR094#SP	AR084#SP
Fn274	CSFO4#SP	CSFO3#SP	CSFO2#SP	CSFO1#SP	FCSS4#SP	FCSS3#SP	FCSS2#SP	FCSS1#SP
Fn275								
Fn276	UO023#P	UO022#P	UO021#P	UO020#P	UO019#P	UO018#P	UO017#P	UO016#P
Fn277	UO031#P	UO030#P	UO029#P	UO028#P	UO027#P	UO026#P	UO025#P	UO024#P
Fn278								
Fn279								
Fn280	UO207#P	UO206#P	UO205#P	UO204#P	UO203#P	UO202#P	UO201#P	UO200#P
Fn281	UO215#P	UO214#P	UO213#P	UO212#P	UO211#P	UO210#P	UO209#P	UO208#P
Fn282	UO223#P	UO222#P	UO221#P	UO220#P	UO219#P	UO218#P	UO217#P	UO216#P
Fn283	UO231#P	UO230#P	UO229#P	UO228#P	UO227#P	UO226#P	UO225#P	UO224#P
Fn284	UO307#P	UO306#P	UO305#P	UO304#P	UO303#P	UO302#P	UO301#P	UO300#P

Address	Bit number 7	6	5	4	3	2	1	0
Fn285	UO315$^{\#P}$	UO314$^{\#P}$	UO313$^{\#P}$	UO312$^{\#P}$	UO311$^{\#P}$	UO310$^{\#P}$	UO309$^{\#P}$	UO308$^{\#P}$
Fn286	UO323$^{\#P}$	UO322$^{\#P}$	UO321$^{\#P}$	UO320$^{\#P}$	UO319$^{\#P}$	UO318$^{\#P}$	UO317$^{\#P}$	UO316$^{\#P}$
Fn287	UO331$^{\#P}$	UO330$^{\#P}$	UO329$^{\#P}$	UO328$^{\#P}$	UO327$^{\#P}$	UO326$^{\#P}$	UO325$^{\#P}$	UO324$^{\#P}$
Fn288					FSPSY4$^{\#SP}$	FSPSY3$^{\#SP}$	FSPSY2$^{\#SP}$	FSPSY1$^{\#SP}$
Fn289					FSPPH4$^{\#SP}$	FSPPH3$^{\#SP}$	FSPPH2$^{\#SP}$	FSPPH1$^{\#SP}$
Fn290			PRGMD$^{\#P}$	PCKSV$^{\#P}$		DNTCM		
Fn291								
Fn292								
Fn293	HPS08$^{\#P}$	HPS07$^{\#P}$	HPS06$^{\#P}$	HPS05$^{\#P}$	HPS04$^{\#P}$	HPS03$^{\#P}$	HPS02$^{\#P}$	HPS01$^{\#P}$
Fn294	HPS16$^{\#P}$	HPS15$^{\#P}$	HPS14$^{\#P}$	HPS13$^{\#P}$	HPS12$^{\#P}$	HPS11$^{\#P}$	HPS10$^{\#P}$	HPS09$^{\#P}$
Fn295	CNCKYO	C2SENO						
Fn296	DM8$^{\#SV}$	DM7$^{\#SV}$	DM6$^{\#SV}$	DM5$^{\#SV}$	DM4$^{\#SV}$	DM3$^{\#SV}$	DM2$^{\#SV}$	DM1$^{\#SV}$
Fn297								MBCAN$^{\#P}$
Fn298	TDSML8$^{\#SV}$	TDSML7$^{\#SV}$	TDSML6$^{\#SV}$	TDSML5$^{\#SV}$	TDSML4$^{\#SV}$	TDSML3$^{\#SV}$	TDSML2$^{\#SV}$	TDSML1$^{\#SV}$
Fn299	TDFTR8$^{\#SV}$	TDFTR7$^{\#SV}$	TDFTR6$^{\#SV}$	TDFTR5$^{\#SV}$	TDFTR4$^{\#SV}$	TDFTR3$^{\#SV}$	TDFTR2$^{\#SV}$	TDFTR1$^{\#SV}$
Fn300								
Fn301								
Fn302								
Fn303								
Fn304								
Fn305								
Fn306				VDCABA				

Address	7	6	5	4	3	2	1	0
Fn307						INESFNA	XPFLA	FRDTEA
Fn308				VDCABB				
Fn309						INESFNB	XPFLB	FRDTEB
Fn310				VDCABC				
Fn311						INESFNC	XPFLC	FRDTEC
Fn312				VDCABD				
Fn313						INESFND	XPFLD	FRDTED
Fn314								
Fn315	TLMEM#P	TMFNFD#P		TLMOT#P		TLMG10#P	TLMSRH#P	TLSKF#P
Fn316	SQMPE#P	SQMPR#P						
Fn317								
Fn318								
Fn319								
Fn320								
Fn321								
Fn322								
Fn323								
Fn324								
Fn325								
Fn326								
Fn327								
Fn328	TLCHI4#P	TLCHI3#P	TLCHI2#P	TLCHI1#P	TLCH4#P	TLCH3#P	TLCH2#P	TLCH1#P

Address	Bit number 7	6	5	4	3	2	1	0
Fn329	TLCHB4#P	TLCHB3#P	TLCHB2#P	TLCHB1#P	TLSKF4#P	TLSKF3#P	TLSKF2#P	TLSKF1#P
Fn330								
Fn331								
Fn332								
Fn333								
Fn334								
Fn335								
Fn336								
Fn337								
Fn338								
Fn339								
Fn340								
Fn341	SYCM8#SV	SYCM7#SV	SYCM6#SV	SYCM5#SV	SYCM4#SV	SYCM3#SV	SYCM2#SV	SYCM1#SV
Fn342	SYCS8#SV	SYCS7#SV	SYCS6#SV	SYCS5#SV	SYCS4#SV	SYCS3#SV	SYCS2#SV	SYCS1#SV
Fn343	MIXO8#SV	MIXO7#SV	MIXO6#SV	MIXO5#SV	MIXO4#SV	MIXO3#SV	MIXO2#SV	MIXO1#SV
Fn344	OVMO8#SV	OVMO7#SV	OVMO6#SV	OVMO5#SV	OVMO4#SV	OVMO3#SV	OVMO2#SV	OVMO1#SV
Fn345	OVSO8#SV	OVSO7#SV	OVSO6#SV	OVSO5#SV	OVSO4#SV	OVSO3#SV	OVSO2#SV	OVSO1#SV
Fn346	SMPK8#SV	SMPK7#SV	SMPK6#SV	SMPK5#SV	SMPK4#SV	SMPK3#SV	SMPK2#SV	SMPK1#SV
Fn347	D3MI#P							
Fn348								
Fn349								
Fn350								
Fn351					SSEGBM4#SP	SSEGBM3#SP	SSEGBM2#SP	SSEGBM1#SP

~

Address	Bit number 7	6	5	4	3	2	1	0
Fn356								
Fn357								
Fn358	WPSF8#SV	WPSF7#SV	WPSF6#SV	WPSF5#SV	WPSF4#SV	WPSF3#SV	WPSF2#SV	WPSF1#SV
~								
Fn376	SVSST8#SV	SVSST7#SV	SVSST6#SV	SVSST5#SV	SVSST4#SV	SVSST3#SV	SVSST2#SV	SVSST1#SV
Fn377	SVSAR8#SV	SVSAR7#SV	SVSAR6#SV	SVSAR5#SV	SVSAR4#SV	SVSAR3#SV	SVSAR2#SV	SVSAR1#SV
~								
Fn381					PHFIND#P	PHFINC#P	PHFINB#P	PHFINA#P
~								
Fn395								
Fn396								
Fn397								
Fn398								
Fn399								
Fn400					SUCLPD#SP	SUCLPC#SP	SUCLPB#SP	
Fn401					SCLPD#SP	SCLPC#SP	SCLPB#SP	
Fn402					MSPOSD#SP	MSPOSC#SP	MSPOSB#SP	
Fn403								SYNER#P
Fn404							COSP4#P	COSP3#P
Fn405								
Fn406								
Fn407								
Fn408								
Fn409								

Address	Bit number 7	6	5	4	3	2	1	0
Fn410								
Fn411								
Fn412								
Fn413								
Fn414								
Fn415								
Fn416								
Fn417								
Fn418								
~								
Fn511								
Fn512						MCSP#P	MCRQ#P	MCEXE#P
Fn513	ZRNR#P		DNCIR#P			MD4R#P	MD2R#P	MD1R#P
Fn514	MCEX8#P	MCEX7#P	MCEX6#P	MCEX5#P	MCEX4#P	MCEX3#P	MCEX2#P	MCEX1#P
Fn515	MCEX16#P	MCEX15#P	MCEX14#P	MCEX13#P	MCEX12#P	MCEX11#P	MCEX10#P	MCEX9#P
Fn516								
Fn517	RP18#SV	RP17#SV	RP16#SV	RP15#SV	RP14#SV	RP13#SV	RP12#SV	RP11#SV
Fn518	RP28#SV	RP27#SV	RP26#SV	RP25#SV	RP24#SV	RP23#SV	RP22#SV	RP21#SV
Fn519								
Fn520					ACDEC#P			ATBK
Fn521	SVREV8#SV	SVREV7#SV	SVREV6#SV	SVREV5#SV	SVREV4#SV	SVREV3#SV	SVREV2#SV	SVREV1#SV
Fn522	SPP8#SV	SPP7#SV	SPP6#SV	SPP5#SV	SPP4#SV	SPP3#SV	SPP2#SV	SPP1#SV
Fn523								
Fn524								

Address	Bit number 7	6	5	4	3	2	1	0
Fn525								
Fn526			DWL$^{\#P}$					
Fn527	SYPER$^{\#P}$	SYPFN$^{\#P}$						
Fn528								
~								
Fn531	IOLBR	DVCPR$^{\#P}$						
Fn532	SYNO8$^{\#SV}$	SYNO7$^{\#SV}$	SYNO6$^{\#SV}$	SYNO5$^{\#SV}$	SYNO4$^{\#SV}$	SYNO3$^{\#SV}$	SYNO2$^{\#SV}$	SYNO1$^{\#SV}$
Fn533								
Fn534	PE3EX	PE2EX	PE1EX	MBSO$^{\#P}$			SRNEX	
Fn535	WFLN2	WFLN1	WETF	WETE	WECCS	WIOCH3	WIOCH2	WIOCH1
Fn536	INIST$^{\#P}$			EXCED$^{\#P}$	ASNED$^{\#P}$	RMVED$^{\#P}$		
~								
Fn540								
Fn541								
Fn542								
Fn543								
Fn544	PSIMF$^{\#P}$						PSIF2$^{\#P}$	PSIF1$^{\#P}$
Fn545				DNTER			OVLNS$^{\#P}$	FLANG
Fn546			GTME$^{\#P}$	GTMC$^{\#P}$				
Fn547	HPMERR	HPMSVM	HPMNTS					
~								
Fn553	FSYSD$^{\#P}$	FSYSC$^{\#P}$	FSYSB$^{\#P}$	FSYSA$^{\#P}$	PHERD$^{\#P}$	PHERC$^{\#P}$	PHERB$^{\#P}$	PHERA$^{\#P}$
Fn554								
~								
Fn558	CDLAD5	CDLAD4	CDPRM	CDDCL	CDLAD3	CDLAD2	CDLAD1	CDCEX
Fn559	SEO8$^{\#SV}$	SEO7$^{\#SV}$	SEO6$^{\#SV}$	SEO5$^{\#SV}$	SEO4$^{\#SV}$	SEO3$^{\#SV}$	SEO2$^{\#SV}$	SEO1$^{\#SV}$

Address	Bit number 7	6	5	4	3	2	1	0
Fn560								
Fn561								
Fn562								
Fn563								
Fn564	M307$^{\#P}$	M306$^{\#P}$	M305$^{\#P}$	M304$^{\#P}$	M303$^{\#P}$	M302$^{\#P}$	M301$^{\#P}$	M300$^{\#P}$
Fn565	M315$^{\#P}$	M314$^{\#P}$	M313$^{\#P}$	M312$^{\#P}$	M311$^{\#P}$	M310$^{\#P}$	M309$^{\#P}$	M308$^{\#P}$
Fn566	M323$^{\#P}$	M322$^{\#P}$	M321$^{\#P}$	M320$^{\#P}$	M319$^{\#P}$	M318$^{\#P}$	M317$^{\#P}$	M316$^{\#P}$
Fn567	M331$^{\#P}$	M330$^{\#P}$	M329$^{\#P}$	M328$^{\#P}$	M327$^{\#P}$	M326$^{\#P}$	M325$^{\#P}$	M324$^{\#P}$
Fn568	M407$^{\#P}$	M406$^{\#P}$	M405$^{\#P}$	M404$^{\#P}$	M403$^{\#P}$	M402$^{\#P}$	M401$^{\#P}$	M400$^{\#P}$
Fn569	M415$^{\#P}$	M414$^{\#P}$	M413$^{\#P}$	M412$^{\#P}$	M411$^{\#P}$	M410$^{\#P}$	M409$^{\#P}$	M408$^{\#P}$
Fn570	M423$^{\#P}$	M422$^{\#P}$	M421$^{\#P}$	M420$^{\#P}$	M419$^{\#P}$	M418$^{\#P}$	M417$^{\#P}$	M416$^{\#P}$
Fn571	M431$^{\#P}$	M430$^{\#P}$	M429$^{\#P}$	M428$^{\#P}$	M427$^{\#P}$	M426$^{\#P}$	M425$^{\#P}$	M424$^{\#P}$
Fn572	M507$^{\#P}$	M506$^{\#P}$	M505$^{\#P}$	M504$^{\#P}$	M503$^{\#P}$	M502$^{\#P}$	M501$^{\#P}$	M500$^{\#P}$
Fn573	M515$^{\#P}$	M514$^{\#P}$	M513$^{\#P}$	M512$^{\#P}$	M511$^{\#P}$	M510$^{\#P}$	M509$^{\#P}$	M508$^{\#P}$
Fn574	M523$^{\#P}$	M522$^{\#P}$	M521$^{\#P}$	M520$^{\#P}$	M519$^{\#P}$	M518$^{\#P}$	M517$^{\#P}$	M516$^{\#P}$
Fn575	M531$^{\#P}$	M530$^{\#P}$	M529$^{\#P}$	M528$^{\#P}$	M527$^{\#P}$	M526$^{\#P}$	M525$^{\#P}$	M524$^{\#P}$
Fn576								
Fn577	SPMER4	SPMER3	SPMER2	SPMER1	SPMFN4	SPMFN3	SPMFN2	SPMFN1
Fn578			ALLO			WBCNT		
Fn579								
Fn580	ARE07$^{\#SP}$	ARE06$^{\#SP}$	ARE05$^{\#SP}$	ARE04$^{\#SP}$	ARE03$^{\#SP}$	ARE02$^{\#SP}$	ARE01$^{\#SP}$	ARE00$^{\#SP}$
Fn581	ARE15$^{\#SP}$	ARE14$^{\#SP}$	ARE13$^{\#SP}$	ARE12$^{\#SP}$	ARE11$^{\#SP}$	ARE10$^{\#SP}$	ARE09$^{\#SP}$	ARE08$^{\#SP}$
Fn582	ARE23$^{\#SP}$	ARE22$^{\#SP}$	ARE21$^{\#SP}$	ARE20$^{\#SP}$	ARE19$^{\#SP}$	ARE18$^{\#SP}$	ARE17$^{\#SP}$	ARE16$^{\#SP}$
Fn583	ARE31$^{\#SP}$	ARE30$^{\#SP}$	ARE29$^{\#SP}$	ARE28$^{\#SP}$	ARE27$^{\#SP}$	ARE26$^{\#SP}$	ARE25$^{\#SP}$	ARE24$^{\#SP}$

Address	Bit number 7	6	5	4	3	2	1	0
Fn584	ARE072$^{\#SP}$	ARE062$^{\#SP}$	ARE052$^{\#SP}$	ARE042$^{\#SP}$	ARE032$^{\#SP}$	ARE022$^{\#SP}$	ARE012$^{\#SP}$	ARE002$^{\#SP}$
Fn585	ARE152$^{\#SP}$	ARE142$^{\#SP}$	ARE132$^{\#SP}$	ARE122$^{\#SP}$	ARE112$^{\#SP}$	ARE102$^{\#SP}$	ARE092$^{\#SP}$	ARE082$^{\#SP}$
Fn586	ARE232$^{\#SP}$	ARE222$^{\#SP}$	ARE212$^{\#SP}$	ARE202$^{\#SP}$	ARE192$^{\#SP}$	ARE182$^{\#SP}$	ARE172$^{\#SP}$	ARE162$^{\#SP}$
Fn587	ARE312$^{\#SP}$	ARE302$^{\#SP}$	ARE292$^{\#SP}$	ARE282$^{\#SP}$	ARE272$^{\#SP}$	ARE262$^{\#SP}$	ARE252$^{\#SP}$	ARE242$^{\#SP}$
Fn588	ARE073$^{\#SP}$	ARE063$^{\#SP}$	ARE053$^{\#SP}$	ARE043$^{\#SP}$	ARE033$^{\#SP}$	ARE023$^{\#SP}$	ARE013$^{\#SP}$	ARE003$^{\#SP}$
Fn589	ARE153$^{\#SP}$	ARE143$^{\#SP}$	ARE133$^{\#SP}$	ARE123$^{\#SP}$	ARE113$^{\#SP}$	ARE103$^{\#SP}$	ARE093$^{\#SP}$	ARE083$^{\#SP}$
Fn590	ARE233$^{\#SP}$	ARE223$^{\#SP}$	ARE213$^{\#SP}$	ARE203$^{\#SP}$	ARE193$^{\#SP}$	ARE183$^{\#SP}$	ARE173$^{\#SP}$	ARE163$^{\#SP}$
Fn591	ARE313$^{\#SP}$	ARE303$^{\#SP}$	ARE293$^{\#SP}$	ARE283$^{\#SP}$	ARE273$^{\#SP}$	ARE263$^{\#SP}$	ARE253$^{\#SP}$	ARE243$^{\#SP}$
Fn592	ARE074$^{\#SP}$	ARE064$^{\#SP}$	ARE054$^{\#SP}$	ARE044$^{\#SP}$	ARE034$^{\#SP}$	ARE024$^{\#SP}$	ARE014$^{\#SP}$	ARE004$^{\#SP}$
Fn593	ARE154$^{\#SP}$	ARE144$^{\#SP}$	ARE134$^{\#SP}$	ARE124$^{\#SP}$	ARE114$^{\#SP}$	ARE104$^{\#SP}$	ARE094$^{\#SP}$	ARE084$^{\#SP}$
Fn594	ARE234$^{\#SP}$	ARE224$^{\#SP}$	ARE214$^{\#SP}$	ARE204$^{\#SP}$	ARE194$^{\#SP}$	ARE184$^{\#SP}$	ARE174$^{\#SP}$	ARE164$^{\#SP}$
Fn595	ARE314$^{\#SP}$	ARE304$^{\#SP}$	ARE294$^{\#SP}$	ARE284$^{\#SP}$	ARE274$^{\#SP}$	ARE264$^{\#SP}$	ARE254$^{\#SP}$	ARE244$^{\#SP}$
~								
Fn598	-OT3O$^{\#P}$	+OT3O$^{\#P}$	-OT2O$^{\#P}$	+OT2O$^{\#P}$	-OT12O$^{\#P}$	+OT12O$^{\#P}$	-OT11O$^{\#P}$	+OT11O$^{\#P}$
Fn599	RTC3O$^{\#P}$	RTC2O$^{\#P}$	CTC3O$^{\#P}$	CTC2O$^{\#P}$	GISO$^{\#P}$			OTSWFN$^{\#P}$
~								
Fn687	SVMWS8$^{\#SV}$	SVMWS7^{SV}	SVMWS6^{SV}	SVMWS5^{SV}	SVMWS4^{SV}	SVMWS3^{SV}	SVMWS2^{SV}	SVMWS1^{SV}
~								
Fn708	RE08O$^{\#SP}$	RE07O$^{\#SP}$	RE06O$^{\#SP}$	RE05O$^{\#SP}$	RE04O$^{\#SP}$	RE03O$^{\#SP}$	RE02O$^{\#SP}$	RE01O$^{\#SP}$
Fn709	RE16O$^{\#SP}$	RE15O$^{\#SP}$	RE14O$^{\#SP}$	RE13O$^{\#SP}$	RE12O$^{\#SP}$	RE11O$^{\#SP}$	RE10O$^{\#SP}$	RE09O$^{\#SP}$
Fn710	RE24O$^{\#SP}$	RE23O$^{\#SP}$	RE22O$^{\#SP}$	RE21O$^{\#SP}$	RE20O$^{\#SP}$	RE19O$^{\#SP}$	RE18O$^{\#SP}$	RE17O$^{\#SP}$
Fn711	RE32O$^{\#SP}$	RE31O$^{\#SP}$	RE30O$^{\#SP}$	RE29O$^{\#SP}$	RE28O$^{\#SP}$	RE27O$^{\#SP}$	RE26O$^{\#SP}$	RE25O$^{\#SP}$
Fn712	RE08O2$^{\#SP}$	RE07O2$^{\#SP}$	RE06O2$^{\#SP}$	RE05O2$^{\#SP}$	RE04O2$^{\#SP}$	RE03O2$^{\#SP}$	RE02O2$^{\#SP}$	RE01O2$^{\#SP}$
Fn713	RE16O2$^{\#SP}$	RE15O2$^{\#SP}$	RE14O2$^{\#SP}$	RE13O2$^{\#SP}$	RE12O2$^{\#SP}$	RE11O2$^{\#SP}$	RE10O2$^{\#SP}$	RE09O2$^{\#SP}$
Fn714	RE24O2$^{\#SP}$	RE23O2$^{\#SP}$	RE22O2$^{\#SP}$	RE21O2$^{\#SP}$	RE20O2$^{\#SP}$	RE19O2$^{\#SP}$	RE18O2$^{\#SP}$	RE17O2$^{\#SP}$

Address	Bit number 7	6	5	4	3	2	1	0
Fn715	RE32O2$^{\#SP}$	RE31O2$^{\#SP}$	RE30O2$^{\#SP}$	RE29O2$^{\#SP}$	RE28O2$^{\#SP}$	RE27O2$^{\#SP}$	RE26O2$^{\#SP}$	RE25O2$^{\#SP}$
Fn716	RE08O3$^{\#SP}$	RE07O3$^{\#SP}$	RE06O3$^{\#SP}$	RE05O3$^{\#SP}$	RE04O3$^{\#SP}$	RE03O3$^{\#SP}$	RE02O3$^{\#SP}$	RE01O3$^{\#SP}$
Fn717	RE16O3$^{\#SP}$	RE15O3$^{\#SP}$	RE14O3$^{\#SP}$	RE13O3$^{\#SP}$	RE12O3$^{\#SP}$	RE11O3$^{\#SP}$	RE10O3$^{\#SP}$	RE09O3$^{\#SP}$
Fn718	RE24O3$^{\#SP}$	RE23O3$^{\#SP}$	RE22O3$^{\#SP}$	RE21O3$^{\#SP}$	RE20O3$^{\#SP}$	RE19O3$^{\#SP}$	RE18O3$^{\#SP}$	RE17O3$^{\#SP}$
Fn719	RE32O3$^{\#SP}$	RE31O3$^{\#SP}$	RE30O3$^{\#SP}$	RE29O3$^{\#SP}$	RE28O3$^{\#SP}$	RE27O3$^{\#SP}$	RE26O3$^{\#SP}$	RE25O3$^{\#SP}$
Fn720	RE08O4$^{\#SP}$	RE07O4$^{\#SP}$	RE06O4$^{\#SP}$	RE05O4$^{\#SP}$	RE04O4$^{\#SP}$	RE03O4$^{\#SP}$	RE02O4$^{\#SP}$	RE01O4$^{\#SP}$
Fn721	RE16O4$^{\#SP}$	RE15O4$^{\#SP}$	RE14O4$^{\#SP}$	RE13O4$^{\#SP}$	RE12O4$^{\#SP}$	RE11O4$^{\#SP}$	RE10O4$^{\#SP}$	RE09O4$^{\#SP}$
Fn722	RE24O4$^{\#SP}$	RE23O4$^{\#SP}$	RE22O4$^{\#SP}$	RE21O4$^{\#SP}$	RE20O4$^{\#SP}$	RE19O4$^{\#SP}$	RE18O4$^{\#SP}$	RE17O4$^{\#SP}$
Fn723	RE32O4$^{\#SP}$	RE31O4$^{\#SP}$	RE30O4$^{\#SP}$	RE29O4$^{\#SP}$	RE28O4$^{\#SP}$	RE27O4$^{\#SP}$	RE26O4$^{\#SP}$	RE25O4$^{\#SP}$
~								
Fn730	TDA8$^{\#SV}$	TDA7$^{\#SV}$	TDA6$^{\#SV}$	TDA5$^{\#SV}$	TDA4$^{\#SV}$	TDA3$^{\#SV}$	TDA2$^{\#SV}$	TDA1$^{\#SV}$
~								
Fn747	TDCF07	TDCF06	TDCF05	TDCF04	TDCF03	TDCF02	TDCF01	TDCF00
~								
Fn767								

A.2 LIST OF SIGNALS

A.2.1 List of Signals (In Order of Functions)

○ : Available
● : Available only with multi path control
- : Unavailable

Function	Signal name	Symbol	Address	T series	M series
2nd feedrate override	2nd feedrate override signals	*AFV0 to *AFV7	G013	○	○
2nd geometry tool offset	Tool offset direction signal	G2RVX	G090.0	○	-
		G2RVZ	G090.1	○	-
		G2RVY	G090.2	○	-
	2nd geometry tool offset axis select signal	G2X	G090.4	○	-
		G2Z	G090.5	○	-
		G2Y	G090.6	○	-
	2nd geometry tool offset signal	G2SLC	G090.7	○	-
2nd reference position return/3rd, 4th reference position return	2nd reference position return completion signals	ZP21 to ZP28	F096	○	○
	3rd reference position return completion signals	ZP31 to ZP38	F098	○	○
	4th reference position return completion signals	ZP41 to ZP48	F100	○	○
Absolute position detection	Absolute position detector battery voltage zero alarm signal	PBATZ	F172.6	○	○
	Absolute position detector battery voltage low alarm signal	PBATL	F172.7	○	○
Actual speed display	Actual speed display axis selection signals	*ACTF1 to *ACTF8	G580	○	○
Actual spindle speed output	Actual spindle speed signals	AR00 to AR15	F040,F041	○	○
		AR002 to AR152	F202,F203	○	○
		AR003 to AR153	F206,F207	○	○
		AR004 to AR154	F272,F273	○	○
AI contour control I/II	AI contour control mode signal	AICC	F062.0	○	○
Alarm signal	Alarm signal	AL	F001.0	○	○
	Battery alarm signal	BAL	F001.2	○	○
Angular axis control	Signal for disabling angular axis control for the perpendicular axis	NOZAGC	G063.5	○	○
Arbitrary speed threading	Chamfering for arbitrary speed threading signal	ASTC	G549.6	○	○
	Groove of thread measurement signal	GTMSR	G549.4	○	○
	Re-machining thread signal	RMTC	G549.5	○	○
	Groove of thread measurement completion signal	GTMC	F546.4	○	○
	Groove of thread measurement error signal	GTME	F546.5	○	○

Function	Signal name	Symbol	Address	T series	M series
Arbitrary spindle position phase synchronization function	Spindle position save start signal	SPMST1 to SPMST4	G587.0 to G587.3	○	○
	Spindle position save selection signal	SMSL11 to SMSL14	G588.0 to G588.3	○	○
		SMSL21 to SMSL24	G588.4 to G588.7	○	○
	Arbitrary spindle position phase synchronization signal	SPAPH1 to SPAPH4	G587.4 to G587.7	○	○
	Spindle position save completion signal	SPMFN1 to SPMFN4	F577.0 to F577.3	○	○
	Spindle position save error signal	SPMER1 to SPMER4	F577.4 to F577.7	○	○
Automatic data backup	Automatic data backup executing signal	ATBK	F520.0	○	○
Automatic phase synchronization for flexible synchronization control	Flexible synchronization control automatic phase synchronization signals	AUTPHA, AUTPHB, AUTPHC, AUTPHD	G381.0 to G381.3	○	○
	Flexible synchronization control phase synchronization end signals	PHFINA, PHFINB, PHFINC, PHFIND	F381.0 to F381.3	○	○
	Automatic phase synchronization error detection signals	PHERA, PHERB, PHERC, PHERD	F553.0 to F553.3	○	○
	Flexible synchronization control mode selecting signals	FSYSA, FSYSB, FSYSC, FSYSD	F553.4 to F553.7	○	○
Automatic tool length measurement (M series)/ Automatic tool offset (T series)	Measuring position reached signals	GAE1	G517.0	○	○
		GAE2	G517.1	○	○
		GAE3	G517.2	-	○
Auxiliary function/2nd auxiliary function	Auxiliary function code signals	M00 to M31	F010 to F013	○	○
	Auxiliary function strobe signals	MF	F007.0	○	○
	Decode M signals	DM00	F009.7	○	○
		DM01	F009.6	○	○
		DM02	F009.5	○	○
		DM30	F009.4	○	○
	Spindle function code signals	S00 to S31	F022 to F025	○	○
Auxiliary function/2nd auxiliary function	Spindle function strobe signal	SF	F007.2	○	○
	Tool function code signals	T00 to T31	F026 to F029	○	○
	Tool function strobe signal	TF	F007.3	○	○
	2nd auxiliary function code signals	B00 to B31	F030 to F033	○	○
	2nd auxiliary function strobe signal	BF	F007.7	○	○
	End signal	FIN	G004.3	○	○
	Distribution completion signals	DEN	F001.3	○	○

Function	Signal name	Symbol	Address	T series	M series
Auxiliary function lock	Auxiliary function lock signal	AFL	G005.6	○	○
	Auxiliary function lock check signal	MAFL	F004.4	○	○
Auxiliary function output block reverse movement for manual handle retrace	Auxiliary function output block reverse movement enable output signal	ADCO	F091.5	○	○
Axis immediate stop	Axis immediate stop start signal	ESTPR	G203.3	○	○
Axis non-display	Axis non-displayed signals	NPOS1 to NPOS8	G198	○	○
Axis switching	Axis switching signals	AXC1	G534.0	○	○
		AXC2	G534.1	○	○
		AXC4	G534.2	○	○
Axis synchronous control	Signals for selecting the manual feed axis for axis synchronous control	SYNCJ1 to SYNCJ8	G140	○	○
	Machine coordinate match state output signals	SYNMT1 to SYNMT8	F210	○	○
	Axis synchronous control status signals	SYNO1 to SYNO8	F532	○	○
	Synchronization compensation enable state output signals	SYNOF1 to SYNOF8	F211	○	○
	Signal for indicating a positional deviation error alarm for axis synchronous control	SYNER	F403.0	○	○
	Synchronous control axis selection signals	SYNC1 to SYNC8	G138	○	○
	Signal for disabling torque difference alarm detection for axis synchronous control	NSYNCA	G059.7	○	○
Axis total travel distance display	Total travel distance clear signal	TDC1 to TDC8	G726	○	○
	Target distance attainment status signal	TDA1 to TDA8	F730	○	○
Canned cycle / multiple repetitive canned cycle	Chamfering signal	*CDZ	G053.7	○	-
Canned cycle for drilling	Small-hole peck drilling cycle in progress signal	PECK2	F066.5	-	○
	Tapping signal	TAP	F001.5	○	○
Changing the display language by pmc signals	Display language switch start signal	SLANG	G581.7	○	○
	Display language setting signals	LANG1 to LANG7	G581.0 to G581.6	○	○
	Display language switch completion signal	FLANG	F545.0	○	○
Chuck / tail stock barrier	Tail stock barrier selection signal	*TSB	G060.7	○	-
CNC ready signal	CNC ready signal	MA	F001.7	○	○
	Servo ready signal	SA	F000.6	○	○
CNC screen dual display	Key control selection signal	CNCKY	G295.7	○	○

Function	Signal name	Symbol	Address	T series	M series
CNC screen dual display	Key control selection status signal	CNCKYO	F295.7	○	○
	Dual display forcible end request signal	C2SEND	G295.6	○	○
	Dual display forcible end status signal	C2SENO	F295.6	○	○
CNC screen Web server function	Web browser connection status signal	WBCNT	F0578.2	○	○
	Web browser connection prohibition signal	WBEND	G0579.5	○	○
Communication Retry Monitoring Function	I/O Link 1 retry abnormality warning signal	WIOCH1	F0535.0	○	○
	I/O Link 2 retry abnormality warning signal	WIOCH2	F0535.1	○	○
	I/O Link 3 retry abnormality warning signal	WIOCH3	F0535.2	○	○
	SRAM ECC abnormality warning signal	WECCS	F0535.3	○	○
	Embedded Ethernet communication abnormality warning signal	WETE	F0535.4	○	○
	Fast Ethernet communication abnormality warning signal	WETF	F0535.5	○	○
	FL-net1 communication abnormality warning signal	WFLN1	F0535.6	○	○
	FL-net2 communication abnormality warning signal	WFLN2	F0535.7	○	○
Constant surface speed control	Constant surface speed signal	CSS	F002.2	○	○
Controlled axis detach	Controlled axis detach signals	DTCH1 to DTCH8	G124	○	○
	Controlled axis detach status signals	MDTCH1 to MDTCH8	F110	○	○
Cs contour control	Cs contour control change signal	CON	G027.7	○	○
	Cs contour control change signal (for each spindle)	CONS1	G274.0	○	○
		CONS2	G274.1	○	○
		CONS3	G274.2	○	○
		CONS4	G274.3	○	○
	Cs contour control change completion signal	FSCSL	F044.1	○	○
	Cs contour control change completion signal (for each spindle)	FCSS1	F274.0	○	○
		FCSS2	F274.1	○	○
		FCSS3	F274.2	○	○
		FCSS4	F274.3	○	○
Cs contour control axis coordinate establishment	Cs axis coordinate establishment request signals	CSFI1	G274.4	○	○
		CSFI2	G274.5	○	○
		CSFI3	G274.6	○	○
		CSFI4	G274.7	○	○
	Cs axis coordinate establishment alarm signals	CSFO1	F274.4	○	○
		CSFO2	F274.5	○	○
		CSFO3	F274.6	○	○
		CSFO4	F274.7	○	○

Function	Signal name	Symbol	Address	T series	M series
Cs contour control axis coordinate establishment	Cs axis origin established state signals	CSPENA	F048.4	O	O
		CSPENB	F052.4	O	O
		CSPENC	F171.4	O	O
		CSPEND	F269.4	O	O
Custom macro	Custom macro input signals	UI000 to UI031	G054 to G057	O	O
		UI100 to UI131	G276 to G279	O	O
		UI200 to UI231	G280 to G283	O	O
		UI300 to UI331	G284 to G287	O	O
	Custom macro output signals	UO000 to UO031	F054,F055, F276,F277	O	O
		UO100 to UO131	F056 to F059	O	O
		UO200 to UO231	F280 to F283	O	O
		UO300 to UO331	F284 to F287	O	O
Cycle start / feed hold	Cycle start signal	ST	G007.2	O	O
	Feed hold signal	*SP	G008.5	O	O
	Automatic operation signal	OP	F000.7	O	O
	Cycle start lamp signal	STL	F000.5	O	O
	Feed hold lamp signal	SPL	F000.4	O	O
Data transfer between PMC and DCSPMC	DI signal for Data transfer between PMC and DCSPMC	TPMG00 to TPMG07	G765	O	O
	DO signal for Data transfer between PMC and DCSPMC	TDCF00 to TDCF07	F747	O	O
DeviceNet Master function	DeviceNet communication normal signal	DNTCM	F290.2	O	O
	DeviceNet communication abnormal signal	DNTER	F545.4	O	O
	DeviceNet communication error clear signal	DNTCLR	G518.4	O	O
Diameter/radius specification switch	Diameter/radius specification switch signals	DI1 to DI8	G296	O	O
	Diameter/radius specification switching in-progress signals	DM1 to DM8	F296	O	O
Direct input of tool offset value measured	Position record signal	PRC	G040.6	O	-
Direct input of tool offset value measured B	Tool offset number selection signals	OFN0 to OFN5, OFN6 to OFN9	G039.0 to G039.5, G040.0 to G040.3	O	-
	Tool offset write mode select signal	GOQSM	G039.7	O	-
	Workpiece coordinate system shift value write mode select signal	WOQSM	G039.6	O	-
	Tool offset write signals	+MIT1,-MIT1 +MIT2,-MIT2	X004.2,X004.3 X004.4,X004.5	O	-
		+MIT1 to +MIT2 -MIT1 to -MIT2	G132.0 to G132.1 G134.0 to G134.1		
	Spindle measurement select signal	S2TLS	G040.5	O	-

Function	Signal name	Symbol	Address	T series	M series
Direct input of tool offset value measured B	Workpiece coordinate system shift value write signal	WOSET	G040.7	○	-
	Tool compensation number specification signal	ONSC	G547.6	○	-
	Spindle 1 under measurement signal	S1MES	F062.3	○	-
	Spindle 2 under measurement signal	S2MES	F062.4	○	-
	Tool offset number selection signals (for milling and turning function)	OFNC0 to OFNC5, OFNC6 to OFNC9	G546.0 to G546.5, G547.0 to G547.3	-	○
	Tool offset write mode select signal (for milling and turning function)	GQSMC	G546.7	-	○
	Tool offset write signals	+MIT1	G132.0	-	○
Direct operation by personal computer function	Direct operation select signal	DMMC	G042.7	○	○
DNC operation	DNC operation select signal	DNCI	G043.5	○	○
	DNC operation selection confirm signal	MRMT	F003.4	○	○
	External device program execution signal	DVCPR	F531.6	○	○
Dry run	Dry run signal	DRN	G046.7	○	○
	Dry run check signal	MDRN	F002.7	○	○
Dual position feedback	Turning mode selection signal	HBTRN	G531.3	○	○
	Compensation clamp signals	*CL1 to *CL8	G548	○	○
Dual control axes switching	Dual control axes switching signal	SVMWC1 to SVMWC8	G687	○	○
	Dual control axes status signal	SVMWS1 to SVMWS8	F687	○	○
Each axis workpiece coordinate system preset signals	Each axis workpiece coordinate system preset signals	WPRST1 to WPRST8	G358	○	○
	Each axis workpiece coordinate system preset completion signals	WPSF1 to WPSF8	F358	○	○
Electronic gear box	Retract signal	RTRCT	G066.4	○	○
	Retract completion signal	RTRCTF	F065.4	○	○
	EGB mode signal	SYNMOD	F065.6	○	○
Electronic gear box 2 pair	EGB synchronization start signals	EGBS1 to EGBS8	G530	○	○
	EGB mode confirmation signals	EGBM1 to EGBM8	F208	○	○
Emergency stop	Emergency stop signals	*ESP	G008.4	○	○
			X008.4, .0, .1	○	○
Extended external machine zero point shift	Extended external machine zero point shift signal	EMZ0 to EMZ15	Specifying by parameter No.1280.	○	○

Function	Signal name	Symbol	Address	T series	M series
External data input	Address signals for external data input	EA6 to EA0	G002.6 to G002.0	○	○
	Data signals for external data input	ED31 to ED0	G211,G210, G001,G000	○	○
	Read signal for external data input	ESTB	G002.7	○	○
	Read completion signal for external data input	EREND	F060.0	○	○
	Search completion signal for external data input	ESEND	F060.1	○	○
	Search cancel signal for external data input	ESCAN	F060.2	○	○
External deceleration	External deceleration signals 1	*+ED1 to *+ED8	G118	○	○
		*-ED1 to *-ED8	G120	○	○
	External deceleration signals 2	*+ED21 to *+ED28	G101	○	○
		*-ED21 to *-ED28	G103	○	○
	External deceleration signals 3	*+ED31 to *+ED38	G107	○	○
		*-ED31 to *-ED38	G109	○	○
	External deceleration signals 4	*+ED41 to *+ED48	G341	○	○
		*-ED41 to *-ED48	G342	○	○
	External deceleration signals 5	*+ED51 to *+ED58	G343	○	○
		*-ED51 to *-ED58	G344	○	○
External I/O device control	External input start signal	EXINP	G058.1	○	○
	External input/output stop signal	EXSTP	G058.2	○	○
	External output start signal	EXOUT	G058.3	○	○
	Input/output busy signal	IOBSY	F053.2	○	○
	Input/output alarm signal	IOALM	F053.3	○	○
	Background editing signal	BGEACT	F053.4	○	○
External key input	External key input mode selection signal	ENBKY	G066.1	○	○
	Key code signals	EKC0 to EKC7	G098	○	○
	Key code read signal	EKSET	G066.7	○	○
	Key code read completion signal	EKENB	F053.7	○	○
	Key input disable signal	INHKY	F053.0	○	○
	Program screen display mode signal	PRGDPL	F053.1	○	○
External workpiece number search	External workpiece number search signals	PN1,PN2,PN4, PN8,PN16	G009.0 to G009.4	○	○
	Extended external workpiece number search signals	EPN0 to EPN13	G024.0 to G025.5	○	○
	External workpiece number search start signal	EPNS	G025.7	○	○
Fan Motor Abnormality Monitoring Function	Alarm level detection signal	SFAN	F093.1	○	○
	Warning level detection signal	WFAN	F093.3	○	○
Feedrate override	Feedrate override signals	*FV0 to *FV7	G012	○	○
Flexible path axis assignment	Removal start signal	RMVST	G536.2	○	○
	Assignment start signal	ASNST	G536.3	○	○
	Exchange start signal	EXCST	G536.4	○	○

Function	Signal name	Symbol	Address	T series	M series
Flexible path axis assignment	Direct assignment mode signal	DASN	G536.5	○	○
	Removal completion signal	RMVED	F536.2	○	○
	Assignment completion signal	ASNED	F536.3	○	○
	Exchange completion signal	EXCED	F536.4	○	○
	Initial axis assignment signal	INIST	F536.7	○	○
Flexible synchronization control	Flexible synchronization control mode selection signals	MTA,MTB,MTC, MTD	G197.0 to G197.3	○	○
	Flexible synchronization control mode status signals	MFSYNA, MFSYNB, MFSYNC, MFSYND	F197.0 to F197.3	○	○
Follow-up	Follow-up signal	*FLWU	G007.5	○	○
Function of deceleration stop in case of power failure	Power failure deceleration signal	PWFL	G203.7	○	○
G code preventing buffering	Speed-up of non-buffering command by G code invalid signal	NHSW	G579.6	○	○
Handle-synchronous feed	Handle-synchronous feed signal	HREV	G023.4	○	○
	Selecting direction of manual handle rotation signal	HDSR	G193.3	○	○
	Feed zero signal	FEED0	F066.2	○	○
High precision oscillation function	Oscillation feedrate override signals	*CHP1 to *CHP8	G051.0 to G051.3	○	○
	Oscillation start signal	CHPST	G051.6	○	○
	Oscillation hold signal	*CHLD	G051.7	○	○
	Oscillation -in-progress signal	CHPMD	F039.2	○	○
	Oscillation cycle signal	CHPCYL	F039.3	○	○
High-speed M/S/T/B interface	Auxiliary function completion signal	MFIN	G005.0	○	○
	Spindle function completion signal	SFIN	G005.2	○	○
	Tool function completion signal	TFIN	G005.3	○	○
	2nd auxiliary function completion signal	BFIN	G005.7	○	○
	2nd M function completion signal	MFIN2	G004.4	○	○
	3rd M function completion signal	MFIN3	G004.5	○	○
	4th M function completion signal	MFIN4	G004.6	○	○
	5th M function completion signal	MFIN5	G004.7	○	○
High-speed position switch	High-speed position switch signals	HPS01 to HPS16	F293,F294	○	○
			Yxxx,Yxxx+1	○	○

Function	Signal name	Symbol	Address	T series	M series
High speed program check	High-speed program check signal	PGCK	G290.5	○	○
	High speed program check mode signal	PRGMD	F290.5	○	○
High speed program check	High speed program check saving data signal	PCKSV	F290.4	○	○
High-speed program management	All programs save request signal	HPMRSV	G514.4	○	○
	Programs not saved status signal	HPMNTS	F547.5	○	○
	Programs saving in progress signal	HPMSVM	F547.6	○	○
	Programs save error signal	HPMERR	F547.7	○	○
High-speed skip	High-speed skip status signals	HDO0 to HDO7	F122	○	○
In-acceleration/ deceleration signal	In-acceleration/ deceleration signal	ACDEC	F520.3	-	○
Inch/metric conversion	Inch input signal	INCH	F002.0	○	○
Index table indexing function	B axis clamp signal	BCLP	F061.1	-	○
	B axis clamp completion signal	*BECLP	G038.7	-	○
	B axis unclamp signal	BUCLP	F061.0	-	○
	B axis unclamp completion signal	*BEUCP	G038.6	-	○
In-feed control (for grinding machine)	In-feed control cut start signal	INFD	G063.6	-	○
In-position check	In-position signals	INP1 to INP8	F104	○	○
	In-position check signal	SMZ	G053.6	○	○
	In-position check disable signal	NOINPS	G023.5	○	○
Interference check	Path interference check association signal	ITF01 to ITF10	G406.0 to G407.1	●	-
	Path interference check in progress signal	TICHK	F064.6	●	-
	Path interference alarm signal	TIALM	F064.7	●	-
Interlock	Start lock signal	STLK	G007.1	○	○
	Interlock signal for all axes	*IT	G008.0	○	○
	Interlock signal for each axis	*IT1 to *IT8	G130	○	○
	Interlock signal for each axis direction	+MIT1 to +MIT8 -MIT1 to -MIT8	G132 G134	-	○
	Cutting block start interlock signal	*CSL	G008.1	○	○
	Block start interlock signal	*BSL	G008.3	○	○
Interrupt type custom macro	Interrupt signal for custom macro	UINT	G053.3	○	○
Inter-path flexible synchronization control	Inter-path flexible synchronization mode select signal	OVLN	G531.4	○	○
	Inter-path flexible synchronization mode signal	OVLNS	F545.1	○	○

Function	Signal name	Symbol	Address	T series	M series
I/O Link βi Manual handle interface (Peripheral equipment control interface)	Manual handle generators selection signal	IOLBH1, IOLBH2	G199.0, G199.1	○	○
	β ready signal	IOLBR	F531.7	○	○
Jog feed/incremental feed	Feed axis and direction selection signals	+J1 to +J8	G100	○	○
		-J1 to -J8	G102	○	○
	Manual feedrate override signals	*JV0 to *JV15	G010,G011	○	○
	Manual rapid traverse selection signal	RT	G019.7	○	○
Machine lock	All-axis machine lock signal	MLK	G044.1	○	○
	Each-axis machine lock signal	MLK1 to MLK8	G108	○	○
	All-axis machine lock check signal	MMLK	F004.1	○	○
Macro executor	Input signals for P-code macro	EUI00 to EUI15	G082,G083	○	○
	Output signals for P-code macro	EUO00 to EUO15	F084,F085	○	○
Manual 2nd/3rd/4th reference position return function	Manual 2nd/3rd/4th reference position return select 1 signal	SLREF	G340.5	○	○
	Manual 2nd/3rd/4th reference position return select 2 signal	SLRER	G340.6	○	○
Manual absolute on/off	Manual absolute signal	*ABSM	G006.2	○	○
	Manual absolute check signal	MABSM	F004.2	○	○
Manual handle feed	Manual handle feed axis selection signals	HS1A to HS1D, HS1E	G018.0 to G018.3, G411.0	○	○
		HS2A to HS2D, HS2E	G018.4 to G018.7, G411.1	○	○
		HS3A to HS3D, HS3E	G019.0 to G019.3, G411.2	○	○
		HS4A to HS4D, HS4E	G020.0 to G020.3, G411.3	○	○
		HS5A to HS5D, HS5E	G379.0 to G379.3, G412.0	○	○
	Manual handle feed amount selection signals (incremental feed signals)	MP1,MP2,MP4	G019.4,G019.5 G019.6	○	○
	Manual handle feed amount selection signals	MP21, MP22 MP31, MP32 MP41, MP42 MP51, MP52	G087.0,G087.1 G087.3,G087.4 G087.6,G087.7 G380.0,G380.1	○	○
	Manual handle feed maximum feedrate change signal	HNDLF	G023.3	○	○
	Manual handle feed direction inversion signal	HDN	G347.1	○	○

Function	Signal name	Symbol	Address	T series	M series
Manual handle interrupt	Manual handle interrupt axis selection signals	HS1IA to HS1ID, HS1IE	G041.0 to G041.3, G411.4	○	○
		HS2IA to HS2ID, HS2IE	G041.4 to G041.7, G411.5	○	○
		HS3IA to HS3ID, HS3IE	G042.0 to G042.3, G411.6	○	○
		HS4IA to HS4ID, HS4IE	G088.4 to G088.7, G411.7	○	○
		HS5IA to HS5ID, HS5IE	G379.4 to G379.7, G412.4	○	○
	3-dimensional coordinate system conversion manual interrupt enable/disable switch signal	NOT3DM	G347.7	○	○
	3-dimensional coordinate conversion manual interrupt mode in-progress signal	D3MI	F347.7	○	○
Manual handle retrace	Checking mode signal	MMOD	G067.2	○	○
	Handle available signal in checking mode	MCHK	G067.3	○	○
	Forward movement prohibition signal	FWSTP	G531.0	○	○
	Reverse movement prohibition signal	MRVM	G531.1	○	○
	Reverse movement signal	MRVMD	F091.0	○	○
	Direction change prohibition signal	MNCHG	F091.1	○	○
	Reverse movement prohibition signal	MRVSP	F091.2	○	○
	Check mode confirmation signal	MMMOD	F091.3	○	○
Manual linear/circular interpolation	Feed axis and direction selection signals	+Jg, -Jg, +Ja, -Ja	G086.0 to G086.3	○	○
	Manual linear/circular interpolation signals	MHLC1 to MHLC5	G544.0 to G544.4	○	○
	Usage selection of manual linear/circular interpolation signals	MHUS1 to MHUS5	G545.0 to G545.4	○	○
Manual reference position return	Manual reference position return selection signal	ZRN	G043.7	○	○
	Manual reference position return selection check signal	MREF	F004.5	○	○
	Reference position return deceleration signals	*DEC1 to *DEC8	G196	○	○
			X009	○	○
	Reference position return end signals	ZP1 to ZP8	F094	○	○
	Reference position establishment signals	ZRF1 to ZRF8	F120	○	○

Function	Signal name	Symbol	Address	T series	M series
Manual tool compensation	Manual tool compensation tool number signal (4 digits)	MTLN00 to MTLN15	G068,G069	O	-
	Manual tool compensation tool number signal (8 digits)	MT8N00 to MT8N31	G525 to G528	O	-
	Manual tool compensation command number	MTLC	G067.0	O	-
	Manual tool compensation completion signal	MTLA	F061.5	O	-
	Manual tool compensation uncompleted signal	MTLANG	F061.4	O	-
Memory protection key	Memory protection signals	KEY1 to KEY4	G046.3 to G046.6	O	O
		KEYP	G046.0	O	O
Mirror image	Mirror image signals	MI1 to MI8	G106	O	O
	Mirror image check signals	MMI1 to MMI8	F108	O	O
Mode selection	Mode selection signals	MD1,MD2,MD4	G043.0 to G043.2	O	O
	Manual data input selection check signal	MMDI	F003.3	O	O
	Automatic operation selection check signal	MMEM	F003.5	O	O
	Program edit selection check signal	MEDT	F003.6	O	O
	Manual handle feed selection check signal	MH	F003.1	O	O
	Incremental feed selection check signal	MINC	F003.0	O	O
	Jog feed selection check signal	MJ	F003.2	O	O
Multiple M commands in a single block	2nd M function code signals	M200 to M215	F014,F015	O	O
		M200 to M231	F014 to F017	O	O
	3rd M function code signals	M300 to M315	F016,F017	O	O
		M300 to M331	F564 to F567	O	O
	4th M function code signals	M400 to M431	F568 to F571	O	O
Multiple M commands in a single block	5th M function code signals	M500 to M531	F572 to F575	O	O
	2nd M function strobe signal	MF2	F008.4	O	O
	3rd M function strobe signal	MF3	F008.5	O	O
	4th M function strobe signal	MF4	F008.6	O	O
	5th M function strobe signal	MF5	F008.7	O	O
Multi-spindle control	Spindle selection signals	SWS1	G027.0	O	O
		SWS2	G027.1	O	O
		SWS3	G027.2	O	O
		SWS4	G026.3	O	O
	Individual spindle stop signals	*SSTP1	G027.3	O	O
		*SSTP2	G027.4	O	O
		*SSTP3	G027.5	O	O

Function	Signal name	Symbol	Address	T series	M series
Multi-spindle control		*SSTP4	G026.6	O	O
	Gear selection signals (input)	GR21,GR22	G029.0,G029.1	O	O
		GR31,GR32	G029.2,G029.3	O	O
		GR41,GR42	G031.4,G031.5	O	O
	2nd position coder selection signal	PC2SLC	G028.7	O	O
	3rd position coder selection signal	PC3SLC	G026.0	O	O
	4th position coder selection signal	PC4SLC	G026.1	O	O
	2nd spindle speed override signals	SOV20 to SOV27	G376	O	O
	3rd spindle speed override signals	SOV30 to SOV37	G377	O	O
	4th spindle speed override signals	SOV40 to SOV47	G378	O	O
	Spindle command path specification signal	SPSP	G536.7	O	O
	Spindle enable signal	ENB2	F038.2	O	O
		ENB3	F038.3	O	O
		ENB4	F039.1	O	O
	S 12-bit code signals	R01O2 to R12O2	F200.0 to F201.3	O	O
		R01O3 to R12O3	F204.0 to F205.3	O	O
		R01O4 to R12O4	F270.0 to F271.3	O	O
	Multi-spindle address P signals	MSP00 to MSP15	F160,F161	O	O
Multi-step skip function	Skip signal	SKIPP	G006.6	O	O
		SKIP	X004.7	O	O
		SKIP2 to SKIP6, SKIP7,SKIP8	X004.2 to X004.6 X004.0,X004.1	O	O
NC data output function	NC data output signal	ALLO	F578.5	O	O
One-digit F code feed	One-digit F code feed signal	F1D	G016.7	-	O
One touch macro call	Macro call start signal	MCST1 to MCST16	G512,G513	O	O
	Mode change completion signal	MCFIN	G514.0	O	O
	Macro call executing signal	MCEXE	F512.0	O	O
	Mode change request signal	MCRQ	F512.1	O	O
	Mode notification signal	MD1R	F513.0	O	O
		MD2R	F513.1	O	O
		MD4R	F513.2	O	O
		DNCIR	F513.5	O	O
		ZRNR	F513.7	O	O
	Abnormal end signal	MCSP	F512.2	O	O
	Call program confirmation signal	MCEX1 to MCEX16	F514,F515	O	O
Operator error prevent	Start check signal	STCHK	G408.0	O	O
	Middle block start signal	MBSO	F534.4	O	O

Function	Signal name	Symbol	Address	T series	M series
Optional block skip/addition of optional block skip	Optional block skip signals	BDT1	G044.0	○	○
		BDT2 to BDT9	G045	○	○
	Optional block skip check signals	MBDT1	F004.0	○	○
		MBDT2 to MBDT9	F005	○	○
Outputting the movement state of an axis	Axis moving signals	MV1 to MV8	F102	○	○
	Axis moving direction signals	MVD1 to MVD8	F106	○	○
Override cancel	Override cancel signal	OVC	G006.4	○	○
Overtravel	Overtravel signals	*+L1 to *+L8	G114	○	○
		*-L1 to *-L8	G116	○	○
Path select	Path select signal (Tool post select signal)	HEAD	G063.0	●	●
	Path select signal 2	HEAD2	G062.7	●	●
	Path select signal 3	HEAD3	G408.1	●	●
	Path select signal 4	HEAD4	G408.2	●	●
Path spindle control	Path spindle command selection signals	SLSPA,SLSPB, SLSPC,SLSPD	G063.2,G063.3, G403.0,G403.1	●	●
	Path spindle feedback selection signals	SLPCA,SLPCB, SLPCC,SLPCD	G064.2,G064.3, G403.4,G403.5	●	●
	Path spindle command confirmation signal	COSP	F064.5	●	●
	Path spindle command confirmation signal	COSP1	F063.3	●	●
		COSP2	F063.4	●	●
		COSP3	F404.0	●	●
		COSP4	F404.1	●	●
Periodic Maintenance Screen	Periodic maintenance lifetime warning signal	LIFOVR	F093.0	○	○
Phase synchronization for Servo/Spindle synchronous control	Phase synchronization for Servo/Spindle synchronous start signal	SYPST	G517.7	○	○
	Phase synchronization for Servo/Spindle synchronous finished signal	SYPFN	F527.6	○	○
	Phase synchronization for Servo/Spindle synchronous error signal	SYPER	F527.7	○	○
PMC axis control / PMC axis speed control function	Control axis selection signals (PMC axis control)	EAX1 to EAX8	G136	○	○
	Axis control command signals (for group 1 to 4) (PMC axis control)	EC0A to EC6A	G143.0 to G143.6	○	○
		EC0B to EC6B	G155.0 to G155.6	○	○
		EC0C to EC6C	G167.0 to G167.6	○	○
		EC0D to EC6D	G179.0 to G179.6	○	○
	Axis control feedrate signals (for group 1 to 4) (PMC axis control)	EIF0A to EIF15A	G144 to G145	○	○
		EIF0B to EIF15B	G156 to G157	○	○
		EIF0C to EIF15C	G168 to G169	○	○
		EIF0D to EIF15D	G180 to G181	○	○

Function	Signal name	Symbol	Address	T series	M series
PMC axis control / PMC axis speed control function	Axis control command read signal (for group 1 to 4) (PMC axis control)	EBUFA	G142.7	○	○
		EBUFB	G154.7	○	○
		EBUFC	G166.7	○	○
		EBUFD	G178.7	○	○
	Axis control data signals (for group 1 to 4) (PMC axis control)	EID0A to EID31A	G146 to G149	○	○
		EID0B to EID31B	G158 to G161	○	○
		EID0C to EID31C	G170 to G173	○	○
		EID0D to EID31D	G182 to G185	○	○
	Axis control command read completion signals (for group 1 to 4) (PMC axis control)	EBSYA	F130.7	○	○
		EBSYB	F133.7	○	○
		EBSYC	F136.7	○	○
		EBSYD	F139.7	○	○
	Reset signal (for group 1 to 4) (PMC axis control)	ECLRA	G142.6	○	○
		ECLRB	G154.6	○	○
		ECLRC	G166.6	○	○
		ECLRD	G178.6	○	○
	Axis control temporary stop signal (for group 1 to 4) (PMC axis control)	ESTPA	G142.5	○	○
		ESTPB	G154.5	○	○
		ESTPC	G166.5	○	○
		ESTPD	G178.5	○	○
	Block stop signal (for group 1 to 4) (PMC axis control)	ESBKA	G142.3	○	○
		ESBKB	G154.3	○	○
		ESBKC	G166.3	○	○
		ESBKD	G178.3	○	○
	Block stop disable signal (for group 1 to 4) (PMC axis control)	EMSBKA	G143.7	○	○
		EMSBKB	G155.7	○	○
		EMSBKC	G167.7	○	○
		EMSBKD	G179.7	○	○
	Auxiliary function code signals (for group 1 to 4) (PMC axis control)	EM11A to EM48A	F132,F142	○	○
		EM11B to EM48B	F135,F145	○	○
		EM11C to EM48C	F138,F148	○	○
		EM11D to EM48D	F141,F151	○	○
	Auxiliary function strobe signal (for group 1 to 4) (PMC axis control)	EMFA	F131.0	○	○
		EMFB	F134.0	○	○
		EMFC	F137.0	○	○
		EMFD	F140.0	○	○
	Auxiliary function 2 strobe signal (for group 1 to 4) (PMC axis control)	EMF2A	F131.2	○	○
		EMF2B	F134.2	○	○
		EMF2C	F137.2	○	○
		EMF2D	F140.2	○	○
	Auxiliary function 3 strobe signal (for group 1 to 4) (PMC axis control)	EMF3A	F131.3	○	○
		EMF3B	F134.3	○	○
		EMF3C	F137.3	○	○
		EMF3D	F140.3	○	○
	Auxiliary function completion signal (for group 1 to 4) (PMC axis control)	EFINA	G142.0	○	○
		EFINB	G154.0	○	○
		EFINC	G166.0	○	○
		EFIND	G178.0	○	○
	Servo-off signal (for group 1 to 4) (PMC axis control)	ESOFA	G142.4	○	○
		ESOFB	G154.4	○	○
		ESOFC	G166.4	○	○
		ESOFD	G178.4	○	○

Function	Signal name	Symbol	Address	T series	M series
PMC axis control / PMC axis speed control function	Buffering disable signal (for group 1 to 4) (PMC axis control)	EMBUFA	G142.2	○	○
		EMBUFB	G154.2	○	○
		EMBUFC	G166.2	○	○
		EMBUFD	G178.2	○	○
	Controlled axis selection status signals (PMC axis control)	*EAXSL	F129.7	○	○
	In-position signal (for group 1 to 4) (PMC axis control)	EINPA	F130.0	○	○
		EINPB	F133.0	○	○
		EINPC	F136.0	○	○
		EINPD	F139.0	○	○
	Following zero checking signals (for group 1 to 4) (PMC axis control)	ECKZA	F130.1	○	○
		ECKZB	F133.1	○	○
		ECKZC	F136.1	○	○
		ECKZD	F139.1	○	○
	Alarm signal (for group 1 to 4) (PMC axis control)	EIALA	F130.2	○	○
		EIALB	F133.2	○	○
		EIALC	F136.2	○	○
		EIALD	F139.2	○	○
	Axis moving signals (for group 1 to 4) (PMC axis control)	EGENA	F130.4	○	○
		EGENB	F133.4	○	○
		EGENC	F136.4	○	○
		EGEND	F139.4	○	○
	Auxiliary function executing signals (for group 1 to 4) (PMC axis control)	EDENA	F130.3	○	○
		EDENB	F133.3	○	○
		EDENC	F136.3	○	○
		EDEND	F139.3	○	○
	Negative-direction overtravel signals (for group 1 to 4) (PMC axis control)	EOTNA	F130.6	○	○
		EOTNB	F133.6	○	○
		EOTNC	F136.6	○	○
		EOTND	F139.6	○	○
	Positive-direction overtravel signals (for group 1 to 4) (PMC axis control)	EOTPA	F130.5	○	○
		EOTPB	F133.5	○	○
		EOTPC	F136.5	○	○
		EOTPD	F139.5	○	○
	Feedrate override signals (for group 1 to 4) (PMC axis control)	*EFOV0 to *EFOV7	G151	○	○
		*EFOV0B to *EFOV7B	G163	○	○
		*EFOV0C to *EFOV7C	G175	○	○
		*EFOV0D to *EFOV7D	G187	○	○
	1% step rapid traverse override signals (for group 1 to 4) (PMC axis control)	*EROV0 to *EROV7	G151	○	○
		*EROV0B to *EROV7B	G163	○	○
		*EROV0C to *EROV7C	G175	○	○
		*EROV0D to *EROV7D	G187	○	○

Function	Signal name	Symbol	Address	T series	M series
PMC axis control / PMC axis speed control function	Override cancellation signal (for group 1 to 4) (PMC axis control)	EOVC	G150.5	○	○
		EOVCB	G162.5	○	○
		EOVCC	G174.5	○	○
		EOVCD	G186.5	○	○
	Rapid traverse override signals (PMC axis control)	EROV1,EROV2	G150.0,G150.1	○	○
	Dry run signal (PMC axis control)	EDRN	G150.7	○	○
	Manual rapid traverse selection signal (PMC axis control)	ERT	G150.6	○	○
	Override 0% signal (PMC axis control)	EOV0	F129.5	○	○
	Skip signal (PMC axis control)	ESKIP	X004.6	○	○
	Distribution completion signals (PMC axis control)	EADEN1 to EADEN8	F112	○	○
	Buffer full signals (for group 1 to 4) (PMC axis control)	EABUFA	F131.1	○	○
		EABUFB	F134.1	○	○
		EABUFC	F137.1	○	○
		EABUFD	F140.1	○	○
	Controlling signals (PMC axis control)	EACNT1 to EACNT8	F182	○	○
	Accumulated zero check signal (for group 1 to 4) (PMC axis control)	ELCKZA	G142.1	○	○
		ELCKZB	G154.1	○	○
		ELCKZC	G166.1	○	○
		ELCKZD	G178.1	○	○
	Torque control mode signal (PMC axis control)	TRQM1 to TRQM8	F190	○	○
	A/B phase detector disconnection alarm ignore signal (PMC axis control)	NDCAL1 to NDCAL8	G202	○	○
	Manual pulse magnification change signal	HNDMP	G088.3	○	○
Polygon turning	Polygon synchronization under way signal	PSYN	F063.7	○	○
Polygon Turning with Two Spindles	Polygon spindle stop signal	*PLSST	G038.0	○	○
	Polygon spindle speed arrival signal	PSAR	F063.2	○	○
	Polygon master axis not arrival signal	PSE1	F063.0	○	○
	Polygon synchronization axis not arrival signal	PSE2	F063.1	○	○
Position switch	Position switch signals	PSW01 to PSW16	F070,F071	○	○
Program restart	Program restart signal	SRN	G006.0	○	○
	Program restart under way signal	SRNMV	F002.4	○	○
	Program restart MDI program output completion signal	SQMPR	F316.6	○	○
	Program restart MDI program execution completion signal	SQMPE	F316.7	○	○

Function	Signal name	Symbol	Address	T series	M series
Quick program restart	Quick program restart under way signal	SRNEX	Fn534.1	O	O
	Program restart memory storing disabled signal	QRSTD	G517.6	O	O
Rapid traverse block overlap	Rapid traverse block overlap disable signal	ROVLP	G053.5	O	O
Rapid traverse override	Rapid traverse override signals	ROV1,ROV2	G014.0,G014.1	O	O
	1% step rapid traverse override selection signals	HROV	G096.7	O	O
	1% rapid traverse override signals	*HROV0 to *HROV6	G096.0 to G096.6	O	O
	0.1% step rapid traverse override selection signal	FHROV	G353.7	O	O
	0.1% rapid traverse override signals	*FHRO0 to *FHRO9	G352.0 to G352.7 G353.0 to G353.1	O	O
Reference point setting with mechanical stopper	Torque limit reach signals for reference point setting with mechanical stopper	CLRCH1 to CLRCH8	F180	O	O
Reference position signal output function	Reference position match signals	RP11 to RP18	F517.0 to F517.7	O	O
	2nd reference position match signals	RP21 to RP28	F518.0 to F518.7	O	O
Reset and rewind	External reset signal	ERS	G008.7	O	O
	MDI reset confirmation signal	MDIRST	F006.1	O	O
	Reset & rewind signal	RRW	G008.6	O	O
	Resetting signal	RST	F001.1	O	O
	Rewinding signal	RWD	F000.0	O	O
	Reset key input invalid signal	IRTKY	G299.7	O	O
Resolution of spindle speed command	S 32-bit code signals	RE01O to RE32O	F708 to F711	O	O
		RE01O2 to RE32O2	F712 to F715	O	O
		RE01O3 to RE32O3	F716 to F719	O	O
		RE01O4 to RE32O4	F720 to F723	O	O
	Extended actual spindle speed signals	ARE00 to ARE31	F580 to F583	O	O
		ARE002 to ARE312	F584 to F587	O	O
		ARE003 to ARE313	F588 to F591	O	O
		ARE004 to ARE314	F592 to F595	O	O
	Extended spindle motor speed command signals	RE01I to RE32I	G708 to G711	O	O
		RE01I2 to RE32I2	G712 to G715	O	O
		RE01I3 to RE32I3	G716 to G719	O	O
		RE01I4 to RE32I4	G720 to G723	O	O

Function	Signal name	Symbol	Address	T series	M series
Retrace	Reverse execution signal	RVS	G007.0	-	○
	Reverse movement signal	RVSL	F082.2	-	○
Retraction for Rigid tapping	Rigid tapping retraction start signal	RTNT	G062.6	○	○
	Rigid tapping retraction completion signal	RTPT	F066.1	○	○
Rigid tapping	Rigid tapping signal	RGTAP	G061.0	○	○
	Spindle rotation direction signals	RGSPP	F065.0	○	○
		RGSPM	F065.1	○	○
	Rigid tapping-in-progress signal	RTAP	F076.3	○	○
	Rigid tapping spindle selection signals	RGTSP1 to RGTSP4	G061.4 to G061.7	○	-
Run hour and part count display	Target part count reached signal	PRTSF	F062.7	○	○
	General-purpose integrating meter start signal	TMRON	G053.0	○	○
Screen erasure	Screen erasure disable signal	*CRTOF	G062.1	○	○
	Automatic screen erasure status in-progress signal	ERTVA	F006.2	○	○
Screen hard copy function	Hard copy cancellation request signal	HCABT	G067.6	○	○
	Hard copy execution request signal	HCREQ	G067.7	○	○
	Hard copy cancellation request reception signal	HCAB2	F061.2	○	○
	Hard copy execution status signal	HCEXE	F061.3	○	○
Servo loop gain / in-position width switching function by signal	Servo loop gain / in-position width switching signal	GIS	G599.3	○	○
	Servo loop gain / in-position width switching confirmation signal	GISO	F599.3	○	○
Servo warning interface	Servo warning detail signals	SVWRN1	F093.4	○	○
		SVWRN2	F093.5	○	○
		SVWRN3	F093.6	○	○
		SVWRN4	F093.7	○	○
Servo off / mechanical handle feed	Servo off signals	SVF1 to SVF8	G126	○	○
Servo/Spindle synchronous control	Servo motor rotation speed specification signals	SVR01I to SVR12I	G021.0 to G022.3	○	○
	Differential speed synchronization command signal	DFSYC	G022.4	○	○
	Servo motor rotation polarity specification signal	SVGN	G022.5	○	○
	Servo motor spindle control switching signal	SVSP	G022.7	○	○
	Servo motor spindle synchronization start signal	SYSS	G061.2	○	○

Function	Signal name	Symbol	Address	T series	M series
Servo/Spindle synchronous control	Servo motor spindle synchronization mode acceleration/deceleration completion signal	SYAR	F090.4	○	○
	Servo motor spindle synchronization mode signal	SYSSM	F090.5	○	○
	Servo motor spindle control mode acceleration/deceleration completion signal	SVAR	F090.6	○	○
	Servo motor spindle control mode signal	SVSPM	F090.7	○	○
Simple spindle electronic gear box	Simple spindle EGB signals	SSEGB1 to SSEGB4	G351.0 to G351.3	○	○
	Simple spindle EGB mode signals	SSEGBM1 to SSEGBM4	F351.0 to F351.3	○	○
Single block	Single block signal	SBK	G046.1	○	○
	Single block check signal	MSBK	F004.3	○	○
Skip function	Skip signal	SKIPP	G006.6	○	○
		SKIP	X004.7	○	○
Software operator's panel	Software operator's panel signal (MD1)	MD1O	F073.0	○	○
	Software operator's panel signal (MD2)	MD2O	F073.1	○	○
	Software operator's panel signal (MD4)	MD4O	F073.2	○	○
	Software operator's panel signal (ZRN)	ZRNO	F073.4	○	○
	Software operator's panel signal (+J1 to +J4)	+J1O to +J4O	F081.0,F081.2, F081.4,F081.6	○	○
	Software operator's panel signal (-J1 to -J4)	-J1O to -J4O	F081.1,F081.3, F081.5,F081.7	○	○
	Software operator's panel signal (RT)	RTO	F077.6	○	○
	Software operator's panel signal (HS1A)	HS1AO	F077.0	○	○
	Software operator's panel signal (HS1B)	HS1BO	F077.1	○	○
	Software operator's panel signal (HS1C)	HS1CO	F077.2	○	○
	Software operator's panel signal (HS1D)	HS1DO	F077.3	○	○
	Software operator's panel signal (MP1)	MP1O	F076.0	○	○
	Software operator's panel signal (MP2)	MP2O	F076.1	○	○
	Software operator's panel signal (*JV0 to *JV15)	*JV0O to *JV15O	F079,F080	○	○
	Software operator's panel signal (*FV0 to *FV7)	*FV0O to *FV7O	F078	○	○
	Software operator's panel signal (ROV1)	ROV1O	F076.4	○	○
	Software operator's panel signal (ROV2)	ROV2O	F076.5	○	○

Function	Signal name	Symbol	Address	T series	M series
Software operator's panel	Software operator's panel signal (BDT)	BDTO	F075.2	○	○
	Software operator's panel signal (SBK)	SBKO	F075.3	○	○
	Software operator's panel signal (MLK)	MLKO	F075.4	○	○
	Software operator's panel signal (DRN)	DRNO	F075.5	○	○
	Software operator's panel signal (KEY1 to KEY4)	KEYO	F075.6	○	○
	Software operator's panel signal (*SP)	SPO	F075.7	○	○
	Software operator's panel general-purpose switch signals	OUT0 to OUT15	F072,F074	○	○
Speed display function of a milling tool with servo motor	Speed display change signal	SDPC	G038.5	○	○
Spindle control mode changing by program command	Spindle control mode off signals	MDOFF1 to MDOFF4	G586.4 to G586.7	○	○
Spindle command synchronous control	Spindle command synchronous control signal	ESRSYC	G064.6	○	○
	Spindle command synchronous control signal (for each spindle)	ESSYC1	G264.0	○	○
		ESSYC2	G264.1	○	○
		ESSYC3	G264.2	○	○
		ESSYC4	G264.3	○	○
	1st spindle parking signal	PKESS1	G122.6 (G031.6)	○	○
	2nd spindle parking signal	PKESS2	G122.7 (G031.7)	○	○
	Spindle command synchronous parking signal (for each spindle)	PKESE1	G265.0	○	○
		PKESE2	G265.1	○	○
		PKESE3	G265.2	○	○
		PKESE4	G265.3	○	○
	Phase error monitor signal	SYCAL	F044.4	○	○
	Phase error monitor signal (for each spindle)	SYCAL1	F043.0	○	○
		SYCAL2	F043.1	○	○
		SYCAL3	F043.2	○	○
		SYCAL4	F043.3	○	○
Spindle control with servo motor	SV speed control mode signals	SRVON1 to SRVON8	G521	○	○
	SV reverse signals	SVRVS1 to SVRVS8	G523	○	○
	SV speed control mode in-progress signals	SVREV1 to SVREV8	F521	○	○
	Spindle indexing signals for each axis	SPP1 to SPP8	F522	○	○
	Speed zero signals	SVSST1 to SVSST8	F376	○	○
	Speed arrival signals	SVSAR1 to SVSAR8	F377	○	○

Function	Signal name	Symbol	Address	T series	M series
Spindle orientation	Spindle orientation signals with the stop position externally set	SH00A to SH14A	G078.0 to G079.6	○	○
		SH00B to SH14B	G080.0 to G081.6	○	○
		SH00C to SH14C	G208.0 to G209.6	○	○
		SH00D to SH14D	G270.0 to G271.6	○	○
Spindle output control by the PMC	Spindle motor speed command selection signals	SIND	G033.7	○	○
		SIND2	G035.7	○	○
		SIND3	G037.7	○	○
		SIND4	G273.7	○	○
	Spindle motor speed command signals	R01I to R12I	G032.0 to G033.3	○	○
		R01I2 to R12I2	G034.0 to G035.3	○	○
		R01I3 to R12I3	G036.0 to G037.3	○	○
		R01I4 to R12I4	G272.0 to G273.3	○	○
	Spindle motor command polarity selection signals	SSIN	G033.6	○	○
		SSIN2	G035.6	○	○
		SSIN3	G037.6	○	○
		SSIN4	G273.6	○	○
	Spindle motor command polarity command signals	SGN	G033.5	○	○
		SGN2	G035.5	○	○
		SGN3	G037.5	○	○
		SGN4	G273.5	○	○
Spindle positioning	Spindle stop complee signal	SPSTPA	G028.6	○	○
		SPSTPB	G402.1	○	○
		SPSTPC	G402.2	○	○
		SPSTPD	G402.3	○	○
	Spindle unclamp signal	SUCLPA	F038.1	○	○
		SUCLPB	F400.1	○	○
		SUCLPC	F400.2	○	○
		SUCLPD	F400.3	○	○
	Spindle unclamp completion signal	*SUCPFA	G028.4	○	○
		*SUCPFB	G400.1	○	○
		*SUCPFC	G400.2	○	○
		*SUCPFD	G400.3	○	○
	Spindle clamp completion signal	*SCPFA	G028.5	○	○
		*SCPFB	G401.1	○	○
		*SCPFC	G401.2	○	○
		*SCPFD	G401.3	○	○
	Spindle clamp signal	SCLPA	F038.0	○	○
		SCLPB	F401.1	○	○
		SCLPC	F401.2	○	○
		SCLPD	F401.3	○	○
	Spindle positioning mode signals	MSPOSA	F039.0	○	○
		MSPOSB	F402.1	○	○
		MSPOSC	F402.2	○	○
		MSPOSD	F402.3	○	○

Function	Signal name	Symbol	Address	T series	M series
Spindle revolution number history function	Total spindle revolution number reset signals	SSR1 to SSR4	G533.0 to G533.3	○	○
	Total spindle revolution number reset selection signal	SSRS	G533.4	○	○
Spindle serial output	Torque limit command LOW signals (serial spindle)	TLMLA	G070.0	○	○
		TLMLB	G074.0	○	○
		TLMLC	G204.0	○	○
		TLMLD	G266.0	○	○
	Torque limit command HIGH signals (serial spindle)	TLMHA	G070.1	○	○
		TLMHB	G074.1	○	○
		TLMHC	G204.1	○	○
		TLMHD	G266.1	○	○
	Clutch/gear signals (serial spindle)	CTH1A,CTH2A	G070.3,G070.2	○	○
		CTH1B,CTH2B	G074.3,G074.2	○	○
		CTH1C,CTH2C	G204.3,G204.2	○	○
		CTH1D,CTH2D	G266.3,G266.2	○	○
	CCW command signals (serial spindle)	SRVA	G070.4	○	○
		SRVB	G074.4	○	○
		SRVC	G204.4	○	○
		SRVD	G266.4	○	○
	CW command signals (serial spindle)	SFRA	G070.5	○	○
		SFRB	G074.5	○	○
		SFRC	G204.5	○	○
		SFRD	G266.5	○	○
	Orientation command signals (serial spindle)	ORCMA	G070.6	○	○
		ORCMB	G074.6	○	○
		ORCMC	G204.6	○	○
		ORCMD	G266.6	○	○
	Machine ready signals (serial spindle)	MRDYA	G070.7	○	○
		MRDYB	G074.7	○	○
		MRDYC	G204.7	○	○
		MRDYD	G266.7	○	○
	Alarm reset signals (serial spindle)	ARSTA	G071.0	○	○
		ARSTB	G075.0	○	○
		ARSTC	G205.0	○	○
		ARSTD	G267.0	○	○
	Emergency stop signals (serial spindle)	*ESPA	G071.1	○	○
		*ESPB	G075.1	○	○
		*ESPC	G205.1	○	○
		*ESPD	G267.1	○	○
	Spindle selection signals (serial spindle)	SPSLA	G071.2	○	○
		SPSLB	G075.2	○	○
		SPSLC	G205.2	○	○
		SPSLD	G267.2	○	○
	Power line switch completion signals (serial spindle)	MCFNA	G071.3	○	○
		MCFNB	G075.3	○	○
		MCFNC	G205.3	○	○
		MCFND	G267.3	○	○
	Soft start/stop signals (serial spindle)	SOCNA	G071.4	○	○
		SOCNB	G075.4	○	○
		SOCNC	G205.4	○	○
		SOCND	G267.4	○	○

Function	Signal name	Symbol	Address	T series	M series
Spindle serial output	Speed integral signals (serial spindle)	INTGA	G071.5	○	○
		INTGB	G075.5	○	○
		INTGC	G205.5	○	○
		INTGD	G267.5	○	○
	Output switch request signals (serial spindle)	RSLA	G071.6	○	○
		RSLB	G075.6	○	○
		RSLC	G205.6	○	○
		RSLD	G267.6	○	○
	Power line status check signals (serial spindle)	RCHA	G071.7	○	○
		RCHB	G075.7	○	○
		RCHC	G205.7	○	○
		RCHD	G267.7	○	○
	Orientation stop position change command signals (serial spindle)	INDXA	G072.0	○	○
		INDXB	G076.0	○	○
		INDXC	G206.0	○	○
		INDXD	G268.0	○	○
	Rotational direction command signals for orientation stop position change (serial spindle)	ROTAA	G072.1	○	○
		ROTAB	G076.1	○	○
		ROTAC	G206.1	○	○
		ROTAD	G268.1	○	○
	Shortcut command signals for orientation stop position change (serial spindle)	NRROA	G072.2	○	○
		NRROB	G076.2	○	○
		NRROC	G206.2	○	○
		NRROD	G268.2	○	○
	Differential speed mode command signals (serial spindle)	DEFMDA	G072.3	○	○
		DEFMDB	G076.3	○	○
		DEFMDC	G206.3	○	○
		DEFMDD	G268.3	○	○
	Analog override signals (serial spindle)	OVRA	G072.4	○	○
		OVRB	G076.4	○	○
		OVRC	G206.4	○	○
		OVRD	G268.4	○	○
	Incremental command externally set orientation signals (serial spindle)	INCMDA	G072.5	○	○
		INCMDB	G076.5	○	○
		INCMDC	G206.5	○	○
		INCMDD	G268.5	○	○
	Spindle switch MAIN MCC contact status signals (serial spindle)	MFNHGA	G072.6	○	○
		MFNHGB	G076.6	○	○
		MFNHGC	G206.6	○	○
		MFNHGD	G268.6	○	○
	Spindle switch HIGH MCC contact status signals (serial spindle)	RCHHGA	G072.7	○	○
		RCHHGB	G076.7	○	○
		RCHHGC	G206.7	○	○
		RCHHGD	G268.7	○	○
	Magnetic sensor orientation command signals (serial spindle)	MORCMA	G073.0	○	○
		MORCMB	G077.0	○	○
		MORCMC	G207.0	○	○
		MORCMD	G269.0	○	○
	Subordinate operation mode command signals (serial spindle)	SLVA	G073.1	○	○
		SLVB	G077.1	○	○
		SLVC	G207.1	○	○
		SLVD	G269.1	○	○

Function	Signal name	Symbol	Address	T series	M series
Spindle serial output	Disconnection detection disable signal (serial spindle)	DSCNA	G073.4	○	○
		DSCNB	G077.4	○	○
		DSCNC	G207.4	○	○
		DSCND	G269.4	○	○
	Inertia estimation start signal (serial spindle)	INESTRA	G304.6	○	○
		INESTRB	G308.6	○	○
		INESTRC	G312.6	○	○
		INESTRD	G316.6	○	○
	Adaptive resonance elimination filter search mode signal (serial spindle)	FRFSMA	G304.7	○	○
		FRFSMB	G308.7	○	○
		FRFSMC	G312.7	○	○
		FRFSMD	G316.7	○	○
	Resonance elimination filter 1 disable signal (serial spindle)	HF1A	G305.0	○	○
		HF1B	G309.0	○	○
		HF1C	G313.0	○	○
		HF1D	G317.0	○	○
	Resonance elimination filter 2 disable signal (serial spindle)	HF2A	G305.1	○	○
		HF2B	G309.1	○	○
		HF2C	G313.1	○	○
		HF2D	G317.1	○	○
	Resonance elimination filter 3 disable signal (serial spindle)	HF3A	G305.2	○	○
		HF3B	G309.2	○	○
		HF3C	G313.2	○	○
		HF3D	G317.2	○	○
	Resonance elimination filter 4 disable signal (serial spindle)	HF4A	G305.3	○	○
		HF4B	G309.3	○	○
		HF4C	G313.3	○	○
		HF4D	G317.3	○	○
	Cutting feed/rapid traverse PWM frequency switching function in Cs contour control enable signal (serial spindle)	PWMSEA	G306.1	○	○
		PWMSEB	G310.1	○	○
		PWMSEC	G314.1	○	○
		PWMSED	G318.1	○	○
	Preload and multi-axis integrator copy disable signal (serial spindle)	TDFCANA	G306.2	○	○
		TDFCANB	G310.2	○	○
		TDFCANC	G314.2	○	○
		TDFCAND	G318.2	○	○
	Motor power cutoff command signals (serial spindle)	MPOFA	G073.2	○	○
		MPOFB	G077.2	○	○
		MPOFC	G207.2	○	○
		MPOFD	G269.2	○	○
	Alarm signals (serial spindle)	ALMA	F045.0	○	○
		ALMB	F049.0	○	○
		ALMC	F168.0	○	○
		ALMD	F266.0	○	○
	Speed zero signals (serial spindle)	SSTA	F045.1	○	○
		SSTB	F049.1	○	○
		SSTC	F168.1	○	○
		SSTD	F266.1	○	○
	Speed detection signals (serial spindle)	SDTA	F045.2	○	○
		SDTB	F049.2	○	○
		SDTC	F168.2	○	○
		SDTD	F266.2	○	○

Function	Signal name	Symbol	Address	T series	M series
Spindle serial output	Spindle speed arrival signal (serial spindle)	SARA	F045.3	○	○
		SARB	F049.3	○	○
		SARC	F168.3	○	○
		SARD	F266.3	○	○
	Load detection signals 1 (serial spindle)	LDT1A	F045.4	○	○
		LDT1B	F049.4	○	○
		LDT1C	F168.4	○	○
		LDT1D	F266.4	○	○
	Load detection signals 2 (serial spindle)	LDT2A	F045.5	○	○
		LDT2B	F049.5	○	○
		LDT2C	F168.5	○	○
		LDT2D	F266.5	○	○
	Torque limit state signals (serial spindle)	TLMA	F045.6	○	○
		TLMB	F049.6	○	○
		TLMC	F168.6	○	○
		TLMD	F266.6	○	○
	Orientation completion signals (serial spindle)	ORARA	F045.7	○	○
		ORARB	F049.7	○	○
		ORARC	F168.7	○	○
		ORARD	F266.7	○	○
	Power line switch signals (serial spindle)	CHPA	F046.0	○	○
		CHPB	F050.0	○	○
		CHPC	F169.0	○	○
		CHPD	F267.0	○	○
	Spindle switch completion signals (serial spindle)	CFINA	F046.1	○	○
		CFINB	F050.1	○	○
		CFINC	F169.1	○	○
		CFIND	F267.1	○	○
	Output switch signals (serial spindle)	RCHPA	F046.2	○	○
		RCHPB	F050.2	○	○
		RCHPC	F169.2	○	○
		RCHPD	F267.2	○	○
	Output switch completion signals (serial spindle)	RCFNA	F046.3	○	○
		RCFNB	F050.3	○	○
		RCFNC	F169.3	○	○
		RCFND	F267.3	○	○
	Subordinate operation status signals (serial spindle)	SLVSA	F046.4	○	○
		SLVSB	F050.4	○	○
		SLVSC	F169.4	○	○
		SLVSD	F267.4	○	○
	Position coder orientation proximity signal (serial spindle)	PORA2A	F046.5	○	○
		PORA2B	F050.5	○	○
		PORA2C	F169.5	○	○
		PORA2D	F267.5	○	○
	Magnetic sensor orientation completion signals (serial spindle)	MORA1A	F046.6	○	○
		MORA1B	F050.6	○	○
		MORA1C	F169.6	○	○
		MORA1D	F267.6	○	○
	Magnetic sensor orientation proximity signals (serial spindle)	MORA2A	F046.7	○	○
		MORA2B	F050.7	○	○
		MORA2C	F169.7	○	○
		MORA2D	F267.7	○	○

Function	Signal name	Symbol	Address	T series	M series
Spindle serial output	Position coder one-rotation signal detection status signals (serial spindle)	PC1DTA	F047.0	○	○
		PC1DTB	F051.0	○	○
		PC1DTC	F170.0	○	○
		PC1DTD	F268.0	○	○
	Incremental orientation mode signals (serial spindle)	INCSTA	F047.1	○	○
		INCSTB	F051.1	○	○
		INCSTC	F170.1	○	○
		INCSTD	F268.1	○	○
	Motor excitation off state signal (serial spindle)	EXOFA	F047.4	○	○
		EXOFB	F051.4	○	○
		EXOFC	F170.4	○	○
		EXOFD	F268.4	○	○
	DC-link failure detection state signal (serial spindle)	VDCABA	F306.4	○	○
		VDCABB	F308.4	○	○
		VDCABC	F310.4	○	○
		VDCABD	F312.4	○	○
	Adaptive resonance elimination filter search completion signal (serial spindle)	FRDTEA	F307.0	○	○
		FRDTEB	F309.0	○	○
		FRDTEC	F311.0	○	○
		FRDTED	F313.0	○	○
	Power failure detection signal (serial spindle)	XPFLA	F307.1	○	○
		XPFLB	F309.1	○	○
		XPFLC	F311.1	○	○
		XPFLD	F313.1	○	○
	Inertia estimation completion signal (serial spindle)	INESFNA	F307.2	○	○
		INESFNB	F309.2	○	○
		INESFNC	F311.2	○	○
		INESFND	F313.2	○	○
	All-spindle operation ready signal	SRSRDY	F034.7	○	○
	1st serial spindle ready signals	SRSP1R	F034.6	○	○
	2nd serial spindle ready signals	SRSP2R	F034.5	○	○
	3rd serial spindle ready signals	SRSP3R	F034.4	○	○
	4th serial spindle ready signals	SRSP4R	F034.3	○	○
Spindle speed control	Spindle warning detail signals 1 to 9	SPWRN1 to SPWRN9	F264.0 to F265.0	○	○
	Spindle stop signal	*SSTP	G029.6	○	○
	Spindle orientation signal	SOR	G029.5	○	○
	Spindle speed override signals	SOV0 to SOV7	G030	○	○
	Spindle speed arrival signal	SAR	G029.4	○	○
	Spindle enable signal	ENB	F001.4	○	○
	Gear selection signals (output)	GR1O,GR2O, GR3O	F034.0 to F034.2	-	○
	Gear selection signals (input)	GR1	G028.1	○	○
		GR2	G028.2	○	○
	S 12-bit code signals	R01O to R12O	F036.0 to F037.3	○	○
Spindle speed fluctuation detection	Spindle speed fluctuation detection alarm signal	SPAL	F035.0	○	○

Function	Signal name	Symbol	Address	T series	M series
Spindle synchronous control	Spindle synchronous control signal	SPSYC	G038.2	○	○
	Spindle synchronous control signal (for each spindle)	SPSYC1	G288.0	○	○
		SPSYC2	G288.1	○	○
		SPSYC3	G288.2	○	○
		SPSYC4	G288.3	○	○
	Spindle phase synchronous control signal	SPPHS	G038.3	○	○
	Spindle phase synchronous control signal (for each spindle)	SPPHS1	G289.0	○	○
		SPPHS2	G289.1	○	○
		SPPHS3	G289.2	○	○
		SPPHS4	G289.3	○	○
	Spindle synchronous speed control completion signal	FSPSY	F044.2	○	○
	Spindle synchronous speed control completion signal (for each spindle)	FSPSY1	F288.0	○	○
		FSPSY2	F288.1	○	○
		FSPSY3	F288.2	○	○
		FSPSY4	F288.3	○	○
	Spindle phase synchronization control completion signal	FSPPH	F044.3	○	○
	Spindle phase synchronization control completion signal (for each spindle)	FSPPH1	F289.0	○	○
		FSPPH2	F289.1	○	○
		FSPPH3	F289.2	○	○
		FSPPH4	F289.3	○	○
	Phase error monitor signal	SYCAL	F044.4	○	○
	Phase error monitor signal (for each spindle)	SYCAL1	F043.0	○	○
		SYCAL2	F043.1	○	○
		SYCAL3	F043.2	○	○
		SYCAL4	F043.3	○	○
	Spindle synchronous speed ratio control clamp signal	RSMAX	F065.2	○	○
	Spindle synchronous speed ratio control signal	SBRT	G038.1	○	○
	Synchronous orientation request command	SORSLA	G073.3	○	○
		SORSLB	G077.3	○	○
		SORSLC	G207.3	○	○
		SORSLD	G269.3	○	○
	Synchronous orientation enable signal	SORENA	F047.3	○	○
		SORENB	F051.3	○	○
		SORENC	F170.3	○	○
		SOREND	F268.3	○	○
Status output signal	Rapid traversing signal	RPDO	F002.1	○	○
	Cutting feed signal	CUT	F002.6	○	○
	Dwell status signal	DWL	F526.5	○	○
Stored stroke check	Stored stroke check 1 switching signals in axis direction	+EXL1 to +EXL8	G104	○	○
		-EXL1 to -EXL8	G105	○	○
	Stored stroke check 1 select signals	EXLM, EXLM2, EXLM3	G007.6, G531.6, G531.7	○	○

Function	Signal name	Symbol	Address	T series	M series
Stored stroke check	Stroke check 1 release signal	RLSOT	G007.7	○	○
	Overtravel alarm signals	+OT1 to +OT8	F124	○	○
		-OT1 to -OT8	F126	○	○
Stored stroke check 2, 3	Stroke check 3 release signal	RLSOT3	G007.4	○	○
Stored stroke limit range switching function by signal	Stored stroke limit range switching data selection signals	OTD0 to OTD15	G594,G595	○	○
	Stored stroke limit range switching axis selection signals	OTA1 to OTA8	G596	○	○
	Stored stroke limit range switching selection signals	+OT11, -OT11, +OT12, -OT12, +OT2, -OT2, +OT3, -OT3	G597	○	○
	Sotred stroke limit range switching cancellation signals	+OT11C, -OT11C, +OT12C, -OT12C, +OT2C, -OT2C, +OT3C, -OT3C	G598	○	○
	Stored stroke limit range switching start signal	OTSW	G599.0	○	○
	Stored stroke limit range switching confirmation signals	+OT11O, -OT11O, +OT12O, -OT12O, +OT2O, -OT2O, +OT3O, -OT3O	F598	○	○
	Stored stroke limit range switching finish signal	OTSWFN	F599.0	○	○
Stroke limit external setting	Stroke limit external setting signals	+LM1 to +LM8	G110	○	○
		-LM1 to -LM8	G112	○	○
Superimposed control	Superimposed control axis selection signals	OVLS1 to OVLS8	G190	○	○
	Superimposed control master axis confirmation signals	OVMO1 to OVMO8	F344	○	○
	Superimposed control slave axis confirmation signals	OVSO1 to OVSO8	F345	○	○
	Synchronous/composite/superimposed control under way signals	SYN1O to SYN8O	F118	○	○
Superimposed control available in the AI contour control mode	Superimposed control axis selection signals	OVLS1 to OVLS8	G190	○	○
	Synchronous/composite/superimposed control under way signals	SYN1O to SYN8O	F118	○	○
	AI contour control permission signal	OVLN	G531.4	○	○
	Advanced superimposition mode signal	OVLNS	F545.1	○	○
Synchronous and composite control	Composite control axis change selection signals	MIX1 to MIX8	G128	○	○
	Composite axis confirmation signals	MIXO1 to MIXO8	F343	○	○

Function	Signal name	Symbol	Address	T series	M series
Synchronous and composite control	Synchronous control axis selection signals	SYNC1 to SYNC8	G138	○	○
	Synchronous master axis confirmation signals	SYCM1 to SYCM8	F341	○	○
	Synchronous slave axis confirmation signals	SYCS1 to SYCS8	F342	○	○
	Synchronous/composite/superimposed control under way signals	SYN1O to SYN8O	F118	○	○
	Parking signals	PK1 to PK8	G122	○	○
	Parking axis confirmation signals	SMPK1 to SMPK8	F346	○	○
	Excess synchronization error signals	SEO1 to SEO8	F559	○	○
Threading	Threading signal	THRD	F002.3	○	○
Three-dimensional coordinate conversion	Three-dimensional coordinate system conversion manual interruption switch signal	M3R	G031.3	○	○
	Three-dimensional coordinate conversion mode signal	D3ROT	F062.6	○	○
Time constant of acceleration / deceleration after interpolation switching function by signal	Time constant of acceleration / deceleration after interpolation for cutting feed switching signals	CTC2 CTC3	G599.4 G599.5	○	○
	Time constant of acceleration / deceleration after interpolation for rapid traverse switching signals	RTC2 RTC3	G599.6 G599.7	○	○
	Time constant of acceleration / deceleration after interpolation for cutting feed switching confirmation signals	CTC2O CTC3O	F599.4 F599.5	○	○
	Time constant of acceleration / deceleration after interpolation for rapid traverse switching confirmation signals	RTC2O RTC3O	F599.6 F599.7	○	○
Tool life management	Tool change signal	TLCH	F064.0	○	○
	Tool change reset signal	TLRST	G048.7	○	○
	Individual tool change signal	TLCHI	F064.2	○	○
	Individual tool change reset signal	TLRSTI	G048.6	○	○
	Tool skip signal	TLSKP	G048.5	○	○
	New tool select signal	TLNW	F064.1	○	○
	Tool group number selection signals	TL01 to TL512	G047.0 to G048.1	○	○
	Tool life count override signals	*TLV0 to *TLV9	G049.0 to G050.1	○	○
	Tool life arrival notice signal	TLCHB	F064.3	○	○

Function	Signal name	Symbol	Address	T series	M series
Tool life management	Tool life counting disable signal	LFCIV	G048.2	○	○
	Tool life counting disabled signal	LFCIF	F093.2	○	○
	Number of remaining tools notification signal	TLAL	F154.0	-	○
Tool management extension function	Tool management data protection signal	TKEY0 to TKEY5	G330.0 to G330.5	○	○
Tool management function	Tool management data modification in-progress signal	TLMG10	F315.2	○	○
	Tool management data output in-progress signal	TLMOT	F315.4	○	○
	Tool management data edit in-progress signal	TLMEM	F315.7	○	○
	Tool search in-progress signal	TLMSRH	F315.1	○	○
	Tool change signal	TLCH	F064.0	○	○
	Tool change signal 1	TLCH1	F328.0	○	○
	Tool change signal 2	TLCH2	F328.1	○	○
	Tool change signal 3	TLCH3	F328.2	○	○
	Tool change signal 4	TLCH4	F328.3	○	○
	Tool change reset signal	TLRST	G048.7	○	○
	Tool change reset signal 1	TLRST1	G328.0	○	○
	Tool change reset signal 2	TLRST2	G328.1	○	○
	Tool change reset signal 3	TLRST3	G328.2	○	○
	Tool change reset signal 4	TLRST4	G328.3	○	○
	Individual tool change signal	TLCHI	F064.2	○	○
	Individual tool change signal 1	TLCHI1	F328.4	○	○
	Individual tool change signal 2	TLCHI2	F328.5	○	○
	Individual tool change signal 3	TLCHI3	F328.6	○	○
	Individual tool change signal 4	TLCHI4	F328.7	○	○
	Individual tool change reset signal	TLRSTI	G048.6	○	○
	Individual tool change reset signal 1	TLRSTI1	G328.4	○	○
	Individual tool change reset signal 2	TLRSTI2	G328.5	○	○
	Individual tool change reset signal 3	TLRSTI3	G328.6	○	○
	Individual tool change reset signal 4	TLRSTI4	G328.7	○	○
	Tool life expiration notice signal	TLCHB	F064.3	○	○
	Tool life expiration notice signal 1	TLCHB1	F329.4	○	○

Function	Signal name	Symbol	Address	T series	M series
Tool management function	Tool life expiration notice signal 2	TLCHB2	F329.5	○	○
	Tool life expiration notice signal 3	TLCHB3	F329.6	○	○
	Tool life expiration notice signal 4	TLCHB4	F329.7	○	○
	Tool skip signal	TLSKP	G048.5	○	○
	Tool skip signal 1	TLSKP1	G329.0	○	○
	Tool skip signal 2	TLSKP2	G329.1	○	○
	Tool skip signal 3	TLSKP3	G329.2	○	○
	Tool skip signal 4	TLSKP4	G329.3	○	○
	Tool skip completion signal	TLSKF	F315.0	○	○
	Tool skip completion signal 1	TLSKF1	F329.0	○	○
	Tool skip completion signal 2	TLSKF2	F329.1	○	○
	Tool skip completion signal 3	TLSKF3	F329.2	○	○
	Tool skip completion signal 4	TLSKF4	F329.3	○	○
	Tool life count override signals	*TLV0 to *TLV9	G049.0 to G050.1	○	○
	Tool life counting disable signal 1	TLNCT1	G329.4	○	○
	Tool life counting disable signal 2	TLNCT2	G329.5	○	○
	Tool life counting disable signal 3	TLNCT3	G329.6	○	○
	Tool life counting disable signal 4	TLNCT4	G329.7	○	○
	Life expiration signal	TMFNFD	F315.6	○	○
Tool retract & recover	Tool retraction axis movement signal	TRMTN	F092.4	○	○
	Tool retraction signal	TRESC	G059.0	○	○
	Tool retraction mode signal	TRACT	F092.3	○	○
	Tool return signal	TRRTN	G059.1	○	○
	Tool return completion signal	TRSPS	F092.5	○	○
Torque limit skip	Torque limit reached signals	TRQL1 to TRQL8	F114	○	○
Touch panel check signal	Touch panel check signal	TPPRS	F006.0	○	○
Trouble diagnosis	Trouble forecast signals for thermal simulation	TDSML1 to TDSML8	F298	○	○
	Trouble forecast signals for disturbance level	TDFTR1 to TDFTR8	F299	○	○
U-axis Control	EGB synchronization mode selection signal	EGBS	G067.4	○	○
	EGB synchronization mode confirmation signal	EGBSM	F082.6	○	○
Unexpected disturbance torque detection	Unexpected disturbance torque detection ignore signals	IUDD1 to IUDD8	G125	○	○

Function	Signal name	Symbol	Address	T series	M series
Unexpected disturbance torque detection	Unexpected disturbance torque detection signals	ABDT1 to ABDT8	F184	○	○
	Servo axis unexpected disturbance torque detection signal	ABTQSV	F090.0	○	○
	1st spindle unexpected disturbance torque detection signal	ABTSP1	F090.1	○	○
	2nd spindle unexpected disturbance torque detection signal	ABTSP2	F090.2	○	○
	3rd spindle unexpected disturbance torque detection signal	ABTSP3	F090.3	○	○
	4th spindle unexpected disturbance torque detection signal	ABTSP4	F091.4	○	○
VRDY off alarm ignore signal	All-axis VRDY off alarm ignore signal	IGNVRY	G066.0	○	○
	Each-axis VRDY off alarm ignore signals	IGVRY1 to IGVRY8	G192	○	○
Waiting M code	No-wait signal	NOWT	G063.1	●	●
	No-wait signal	NMWT	G063.7	●	●
	Waiting signal	WATO	F063.6	●	●
Waiting M codes of high-speed type	Waiting M codes of high-speed type invalid signal	NHSW	G579.6	●	●
Warning function against modification of setting	Notification signal for modification of C Language Executor program	CDCEX	F558.0	○	○
	Notification signal for modification of 1st path PMC Ladder program	CDLAD1	F558.1	○	○
	Notification signal for modification of 2nd path PMC Ladder program	CDLAD2	F558.2	○	○
	Notification signal for modification of 3rd path PMC Ladder program	CDLAD3	F558.3	○	○
	Notification signal for modification of Dual Check Safety PMC Ladder program	CDDCL	F558.4	○	○
	Notification signal for modification of CNC parameter	CDPRM	F558.5	○	○
	Notification signal for modification of 4th path PMC Ladder program	CDLAD4	F558.6	○	○
	Notification signal for modification of 5th path PMC Ladder program	CDLAD5	F558.7	○	○

A.2.2 List of Signals (In Order of Symbols)

○ : Available
● : Available only with multi path control
- : Unavailable

Group	Symbol	Signal name	Address	T series	M series
*	*+ED1 to *+ED8	External deceleration signals 1	G118	○	○
	*+ED21 to *+ED28	External deceleration signals 2	G101	○	○
	*+ED31 to *+ED38	External deceleration signals 3	G107	○	○
	*+ED41 to *+ED48	External deceleration signals 4	G341	○	○
	*+ED51 to *+ED58	External deceleration signals 5	G343	○	○
	*+L1 to *+L8	Overtravel signals	G114	○	○
	*-ED1 to *-ED8	External deceleration signals 1	G120	○	○
	*-ED21 to *-ED28	External deceleration signals 2	G103	○	○
	*-ED31 to *-ED38	External deceleration signals 3	G109	○	○
	*-ED41 to *-ED48	External deceleration signals 4	G342	○	○
	*-ED51 to *-ED58	External deceleration signals 5	G344	○	○
	*-L1 to *-L8	Overtravel signals	G116	○	○
	*ABSM	Manual absolute signal	G006.2	○	○
	*ACTF1 to *ACTF8	Actual speed display axis selection signals	G580	○	○
	*AFV0 to *AFV7	2nd feedrate override signals	G013	○	○
	*BSL	Block start interlock signal	G008.3	○	○
	*BECLP	B axis clamp completion signal	G038.7	-	○
	*BEUCP	B axis unclamp completion signal	G038.6	-	○
	*CDZ	Chamfering signal	G053.7	○	-
	*CHLD	Chopping/oscillation hold signal	G051.7	○	○
	*CHP1 to *CHP8	Chopping/oscillation feedrate override signals	G051.0 to G051.3	○	○
	*CL1 to *CL8	Dual position feedback compensation clamp signals	G548	○	○
	*CRTOF	Screen erasure disable signal	G062.1	○	○
	*CSL	Cutting block start interlock signal	G008.1	○	○
	*DEC1 to *DEC8	Reference position return deceleration signals	X009	○	○
			G196	○	○
	*EAXSL	Controlled axis selection status signals (PMC axis control)	F129.7	○	○
	*ESP	Emergency stop signals	X008.4, .0, .1	○	○
			G008.4	○	○
	*ESPA	Emergency stop signals (serial spindle)	G071.1	○	○
	*ESPB		G075.1	○	○
	*ESPC		G205.1	○	○
	*ESPD		G267.1	○	○
	*EFOV0 to *EFOV7	Feedrate override signals (for group 1 to 4) (PMC axis control)	G151	○	○
	*EFOV0B to *EFOV7B		G163	○	○
	*EFOV0C to *EFOV7C		G175	○	○
	*EFOV0D to *EFOV7D		G187	○	○
	*EROV0 to *EROV7	1% step rapid traverse override signals (for group 1 to 4) (PMC axis control)	G151	○	○
	*EROV0B to *EROV7B		G163	○	○
	*EROV0C to *EROV7C		G175	○	○
	*EROV0D to *EROV7D		G187	○	○

Group	Symbol	Signal name	Address	T series	M series
*	*FHRO0 to *FHRO9	0.1% rapid traverse override signals	G352.0 to G352.7 G353.0 to G353.1	○	○
	*FLWU	Follow-up signal	G007.5	○	○
	*FV0 to *FV7	Feedrate override signals	G012	○	○
	*FV0O to *FV7O	Software operator’s panel signal (*FV0 to *FV7)	F078	○	○
	*HROV0 to *HROV6	1% rapid traverse override signals	G096.0 to G096.6	○	○
	*IT	Interlock signal for all axes	G008.0	○	○
	*IT1 to *IT8	Interlock signal for each axis	G130	○	○
	*JV0 to *JV15	Manual feedrate override signals	G010,G011	○	○
	*JV0O to *JV15O	Software operator’s panel signal (*JV0 to *JV15)	F079,F080	○	○
	*PLSST	Polygon spindle stop signal	G038.0	○	○
	*SCPFA	Spindle clamp completion signal	G028.5	○	○
	*SCPFB		G401.1	○	○
	*SCPFC		G401.2	○	○
	*SCPFD		G401.3	○	○
	*SP	Feed hold signal	G008.5	○	○
	*SSTP	Spindle stop signal	G029.6	○	○
	*SSTP1	Individual spindle stop signals	G027.3	○	○
	*SSTP2		G027.4	○	○
	*SSTP3		G027.5	○	○
	*SSTP4		G026.6	○	○
	*SUCPFA	Spindle unclamp completion signal	G028.4	○	○
	*SUCPFB		G400.1	○	○
	*SUCPFC		G400.2	○	○
	*SUCPFD		G400.3	○	○
	*TLV0 to *TLV9	Tool life count override signals	G049.0 to G050.1	○	○
	*TSB	Tail stock barrier selection signal	G060.7	○	-
+	+EXL1 to +EXL8	Stored stroke check 1 switching signals in axis direction	G104	○	○
	+J1 to +J8	Feed axis and direction selection signals	G100	○	○
	+J1O to +J4O	Software operator’s panel signal (+J1 to +J4)	F081.0,F081.2, F081.4,F081.6	○	○
	+Jg,+Ja	Feed axis and direction selection signals	G086.0,G086.2	○	○
	+LM1 to +LM8	Stroke limit external setting signals	G110	○	○
	+MIT1,+MIT2	Tool offset write signals	X004.2,X004.4 G132.0,G132.1	○	-
	+MIT1	Tool offset write signals	G132.0	-	○
	+MIT1 to +MIT8	Interlock signal for each axis direction	G132	-	○
	+OT1 to +OT8	Overtravel alarm signals	F124	○	○
	+OT11	Stored stroke limit range switching selection signals	G597.0	○	○
	+OT12		G597.2	○	○
	+OT2		G597.4	○	○
	+OT3		G597.6	○	○

Group	Symbol	Signal name	Address	T series	M series
+	+OT11C	Stored stroke limit range switching cancellation signals	G598.0	○	○
	+OT12C		G598.2	○	○
	+OT2C		G598.4	○	○
	+OT3C		G598.6	○	○
	+OT11O	Stored stroke limit range switching confirmation signals	F598.0	○	○
	+OT12O		F598.2	○	○
	+OT2O		F598.4	○	○
	+OT3O		F598.6	○	○
-	-EXL1 to -EXL8	Stored stroke check 1 switching signals in axis direction	G105	○	○
	-J1 to -J8	Feed axis and direction selection signals	G102	○	○
	-J1O to -J4O	Software operator's panel signal (-J1 to -J4)	F081.1,F081.3, F081.5,F081.7	○	○
	-Jg,-Ja	Feed axis and direction selection signals	G086.1,G086.3	○	○
	-LM1 to -LM8	Stroke limit external setting signals	G112	○	○
	-MIT1,-MIT2	Tool offset write signals	X004.3,X004.5 G134.0,G134.1	○	-
	-MIT1 to -MIT8	Interlock signal for each axis direction	G134	-	○
	-OT1 to -OT8	Overtravel alarm signals	F126	○	○
	-OT11	Stored stroke limit range switching selection signals	G597.1	○	○
	-OT12		G597.3	○	○
	-OT2		G597.5	○	○
	-OT3		G597.7	○	○
	-OT11C	Stored stroke limit range switching cancellation signals	G598.1	○	○
	-OT12C		G598.3	○	○
	-OT2C		G598.5	○	○
	-OT3C		G598.7	○	○
	-OT11O	Stored stroke limit range switching confirmation signals	F598.1	○	○
	-OT12O		F598.3	○	○
	-OT2O		F598.5	○	○
	-OT3O		F598.7	○	○
A	ABDT1 to ABDT8	Unexpected disturbance torque detection signal	F184	○	○
	ABTQSV	Servo axis unexpected disturbance torque detection signal	F090.0	○	○
	ABTSP1	1st spindle unexpected disturbance torque detection signal	F090.1	○	○
	ABTSP2	2nd spindle unexpected disturbance torque detection signal	F090.2	○	○
	ABTSP3	3rd spindle unexpected disturbance torque detection signal	F090.3	○	○
	ABTSP4	4th spindle unexpected disturbance torque detection signal	F091.4	○	○

Group	Symbol	Signal name	Address	T series	M series
A	ACDEC	In-acceleration/deceleration signal	F520.3	-	○
	ADCO	Auxiliary function output block reverse movement enable output signal	F091.5	○	○
	AFL	Auxiliary function lock signal	G005.6	○	○
	AICC	AI contour control mode signal	F062.0	○	○
	AL	Alarm signal	F001.0	○	○
	ALLO	NC data output signal	F578.5	○	○
	ALMA	Alarm signals (serial spindle)	F045.0	○	○
	ALMB		F049.0	○	○
	ALMC		F168.0	○	○
	ALMD		F266.0	○	○
	ALNGH	Tool axis direction feed mode signal	G023.7	○	○
	AR00 to AR15	Actual spindle speed signals	F040,F041	○	○
	AR002 to AR152		F202,F203	○	○
	AR003 to AR153		F206,F207	○	○
	AR004 to AR154		F272,F273	○	○
	ARE00 to ARE31	Extended actual spindle speed signals	F580 to F583	○	○
	ARE002 to ARE312		F584 to F587	○	○
	ARE003 to ARE313		F588 to F591	○	○
	ARE004 to ARE314		F592 to F595	○	○
	ARSTA	Alarm reset signals (serial spindle)	G071.0	○	○
	ARSTB		G075.0	○	○
	ARSTC		G205.0	○	○
	ARSTD		G267.0	○	○
	ASNST	Assignment start signal	G536.3	○	○
	ASNED	Assignment completion signal	F536.3	○	○
	ASTC	Chamfering for arbitrary speed threading signal	G549.6	○	○
	ATBK	Automatic data backup executing signal	F520.0	○	○
	AUTPHA to AUTPHD	Flexible synchronization control automatic phase synchronization signals	G381.0 to G381.3	○	○
	AXC1	Axis switching signals	G534.0	○	○
	AXC2		G534.1	○	○
	AXC4		G534.2	○	○
B	B00 to B31	2nd auxiliary function code signals	F030 to F033	○	○
	BAL	Battery alarm signal	F001.2	○	○
	BCAN	Block cancel signal	G297.0	○	○
	BCLP	B axis clamp signal	F061.1	-	○
	BDT1	Optional block skip signals	G044.0	○	○
	BDT2 to BDT9		G045	○	○
	BDTO	Software operator’s panel signal (BDT)	F075.2	○	○
	BF	2nd auxiliary function strobe signal	F007.7	○	○
	BFIN	2nd auxiliary function completion signal	G005.7	○	○
	BGEACT	Background editing signal	F053.4	○	○

Group	Symbol	Signal name	Address	T series	M series
B	BOV	High-speed cycle machining override selection signal	G518.0	○	○
	BUCLP	B axis unclamp signal	F061.0	-	○
C	C2SEND	Dual display forcible end request signal	G295.6	○	○
	C2SENO	Dual display forcible end status signal	F295.6	○	○
	CDCEX	Notification signal for modification of C Language Executor program	F558.0	○	○
	CDDCL	Notification signal for modification of Dual Check Safety PMC Ladder program	F558.4	○	○
	CDLAD1	Notification signal for modification of 1st path PMC Ladder program	F558.1	○	○
	CDLAD2	Notification signal for modification of 2nd path PMC Ladder program	F558.2	○	○
	CDLAD3	Notification signal for modification of 3rd path PMC Ladder program	F558.3	○	○
	CDLAD4	Notification signal for modification of 4th path PMC Ladder program	F558.6	○	○
	CDLAD5	Notification signal for modification of 5th path PMC Ladder program	F558.7	○	○
	CDPRM	Notification signal for modification of CNC parameter	F558.5	○	○
	CFINA	Spindle switch completion signals (serial spindle)	F046.1	○	○
	CFINB		F050.1	○	○
	CFINC		F169.1	○	○
	CFIND		F267.1	○	○
	CHPA	Power line switch signals (serial spindle)	F046.0	○	○
	CHPB		F050.0	○	○
	CHPC		F169.0	○	○
	CHPD		F267.0	○	○
	CHPCYL	Oscillation cycle signal	F039.3	○	○
	CHPMD	Oscillation -in-progress signal	F039.2	○	○
	CHPST	Oscillation start signal	G051.6	○	○
	CLRCH1 to CLRCH8	Torque limit reach signals for reference pointsetting with mechanical stopper	F180	○	○
	CNCKY	Key control selection signal	G295.7	○	○
	CNCKYO	Key control selection status signal	F295.7	○	○
	CON	Cs contour control change signal	G027.7	○	○
	CONH1	Cs contour control high speed switching signals	G549.0	○	○
	CONH2		G549.1	○	○
	CONH3		G549.2	○	○
	CONH4		G549.3	○	○

Group	Symbol	Signal name	Address	T series	M series
C	CONS1	Cs contour control change signal (for each spindle)	G274.0	○	○
	CONS2		G274.1	○	○
	CONS3		G274.2	○	○
	CONS4		G274.3	○	○
	COSP	Path spindle command confirmation signal	F064.5	●	●
	COSP1	Path spindle command confirmation signal	F063.3	●	●
	COSP2		F063.4	●	●
	COSP3		F404.0	●	●
	COSP4		F404.1	●	●
	CS1DTA	One-rotation signal detection status signal for Cs contour control (serial spindle)	F047.6	○	○
	CS1DTB		F051.6	○	○
	CS1DTC		F170.6	○	○
	CS1DTD		F268.6	○	○
	CSFI1	Cs axis coordinate establishment request signals	G274.4	○	○
	CSFI2		G274.5	○	○
	CSFI3		G274.6	○	○
	CSFI4		G274.7	○	○
	CSFO1	Cs axis coordinate establishment alarm signals	F274.4	○	○
	CSFO2		F274.5	○	○
	CSFO3		F274.6	○	○
	CSFO4		F274.7	○	○
	CSMC1	Cs contour control high speed switching completion signals	F546.0	○	○
	CSMC2		F546.1	○	○
	CSMC3		F546.2	○	○
	CSMC4		F546.3	○	○
	CSPENA	Cs axis origin established state signals	F048.4	○	○
	CSPENB		F052.4	○	○
	CSPENC		F171.4	○	○
	CSPEND		F269.4	○	○
	CSS	Constant surface speed signal	F002.2	○	○
	CSYCA	Reference position establishment starting signal (serial spindle)	G304.3	○	○
	CSYCB		G308.3	○	○
	CSYCC		G312.3	○	○
	CSYCD		G316.3	○	○
	CSYFNA	Reference position establishment completion signal (serial spindle)	F047.7	○	○
	CSYFNB		F051.7	○	○
	CSYFNC		F170.7	○	○
	CSYFND		F268.7	○	○
	CTC2	Time constant of acceleration / deceleration after interpolation for cutting feed switching signals	G599.4	○	○
	CTC3		G599.5	○	○
	CTC2O	Time constant of acceleration / deceleration after interpolation for cutting feed switching confirmation signals	F599.4	○	○
	CTC3O		F599.5	○	○
	CTH1A,CTH2A	Clutch/gear signals (serial spindle)	G070.3,G070.2	○	○
	CTH1B,CTH2B		G074.3,G074.2	○	○
	CTH1C,CTH2C		G204.3,G204.2	○	○
	CTH1D,CTH2D		G266.3,G266.2	○	○
	CUT	Cutting feed signal	F002.6	○	○

Group	Symbol	Signal name	Address	T series	M series
D	D3MI	3-dimensional coordinate conversion manual interrupt mode in-progress signal	F347.7	○	○
	D3ROT	Three-dimensional coordinate conversion mode signal	F062.6	○	○
	DASN	Direct assignment mode signal	G536.5	○	○
	DEFMDA	Differential speed mode command signals (serial spindle)	G072.3	○	○
	DEFMDB		G076.3	○	○
	DEFMDC		G206.3	○	○
	DEFMDD		G268.3	○	○
	DEN	Distribution completion signals	F001.3	○	○
	DFSYC	Differential speed synchronization command signal	G022.4	○	○
	DI1 to DI8	Diameter/radius specification switch signals	G296	○	○
	DM1 to DM8	Diameter/radius specification switching in-progress signals	F296	○	○
	DM00	Decode M signals	F009.7	○	○
	DM01		F009.6	○	○
	DM02		F009.5	○	○
	DM30		F009.4	○	○
	DMMC	Direct operation select signal	G042.7	○	○
	DNCI	DNC operation select signal	G043.5	○	○
	DNCIR	Mode notification signal	F513.5	○	○
	DNTCLR	DeviceNet communication error clear signal	G518.4	○	○
	DNTCM	DeviceNet communication normal signal	F290.2	○	○
	DNTER	DeviceNet communication abnormal signal	F545.4	○	○
	DRN	Dry run signal	G046.7	○	○
	DRNO	Software operator's panel signal (DRN)	F075.5	○	○
	DSCNA	Disconnection detection disable signal (serial spindle)	G073.4	○	○
	DSCNB		G077.4	○	○
	DSCNC		G207.4	○	○
	DSCND		G269.4	○	○
	DTCH1 to DTCH8	Controlled axis detach signals	G124	○	○
	DVCPR	External device program execution signal	F531.6	○	○
	DWL	Dwell status signal	F526.5	○	○
E	EA6 to EA0	Address signals for external data input	G002.6 to G002.0	○	○
	EABUFA	Buffer full signals (for group 1 to 4) (PMC axis control)	F131.1	○	○
	EABUFB		F134.1	○	○
	EABUFC		F137.1	○	○
	EABUFD		F140.1	○	○
	EACNT1 to EACNT8	Controlling signals (PMC axis control)	F182	○	○
	EADEN1 to EADEN8	Distribution completion signals (PMC axis control)	F112	○	○
	EAX1 to EAX8	Control axis selection signals (PMC axis control)	G136	○	○

Group	Symbol	Signal name	Address	T series	M series
E	EBSYA	Axis control command read completion signals (for group 1 to 4) (PMC axis control)	F130.7	○	○
	EBSYB		F133.7	○	○
	EBSYC		F136.7	○	○
	EBSYD		F139.7	○	○
	EBUFA	Axis control command read signal (for group 1 to 4) (PMC axis control)	G142.7	○	○
	EBUFB		G154.7	○	○
	EBUFC		G166.7	○	○
	EBUFD		G178.7	○	○
	EC0A to EC6A	Axis control command signals (for group 1 to 4) (PMC axis control)	G143.0 to G143.6	○	○
	EC0B to EC6B		G155.0 to G155.6	○	○
	EC0C to EC6C		G167.0 to G167.6	○	○
	EC0D to EC6D		G179.0 to G179.6	○	○
	ECKZA	Following zero checking signals (for group 1 to 4) (PMC axis control)	F130.1	○	○
	ECKZB		F133.1	○	○
	ECKZC		F136.1	○	○
	ECKZD		F139.1	○	○
	ECLRA	Reset signal (for group 1 to 4) (PMC axis control)	G142.6	○	○
	ECLRB		G154.6	○	○
	ECLRC		G166.6	○	○
	ECLRD		G178.6	○	○
	ED31 to ED0	Data signals for external data input	G211,G210, G001,G000	○	○
	EDENA	Auxiliary function executing signals (for group 1 to 4) (PMC axis control)	F130.3	○	○
	EDENB		F133.3	○	○
	EDENC		F136.3	○	○
	EDEND		F139.3	○	○
	EDRN	Dry run signal (PMC axis control)	G150.7	○	○
	EFINA	Auxiliary function completion signal (for group 1 to 4) (PMC axis control)	G142.0	○	○
	EFINB		G154.0	○	○
	EFINC		G166.0	○	○
	EFIND		G178.0	○	○
	EGENA	Axis moving signals (for group 1 to 4) (PMC axis control)	F130.4	○	○
	EGENB		F133.4	○	○
	EGENC		F136.4	○	○
	EGEND		F139.4	○	○
	EGBM1 to EGBM8	EGB mode confirmation signals	F208	○	○
	EGBS	EGB synchronization mode selection signal	G067.4	○	○
	EGBS1 to EGBS8	EGB synchronization start signals	G530	○	○
	EGBSM	EGB synchronization mode confirmation signal	F082.6	○	○
	EIALA	Alarm signal (for group 1 to 4) (PMC axis control)	F130.2	○	○
	EIALB		F133.2	○	○
	EIALC		F136.2	○	○
	EIALD		F139.2	○	○
	EID0A to EID31A	Axis control data signals (for group 1 to 4) (PMC axis control)	G146 to G149	○	○
	EID0B to EID31B		G158 to G161	○	○
	EID0C to EID31C		G170 to G173	○	○
	EID0D to EID31D		G182 to G185	○	○

Group	Symbol	Signal name	Address	T series	M series
E	EIF0A to EIF15A	Axis control feedrate signals (for group 1 to 4) (PMC axis control)	G144,G145	○	○
	EIF0B to EIF15B		G156,G157	○	○
	EIF0C to EIF15C		G168,G169	○	○
	EIF0D to EIF15D		G180,G181	○	○
	EINPA	In-position signal (for group 1 to 4) (PMC axis control)	F130.0	○	○
	EINPB		F133.0	○	○
	EINPC		F136.0	○	○
	EINPD		F139.0	○	○
	EKC0 to EKC7	Key code signals	G098	○	○
	EKENB	Key code read completion signal	F053.7	○	○
	EKSET	Key code read signal	G066.7	○	○
	ELCKZA	Accumulated zero check signal (for group 1 to 4) (PMC axis control)	G142.1	○	○
	ELCKZB		G154.1	○	○
	ELCKZC		G166.1	○	○
	ELCKZD		G178.1	○	○
	EM11A to EM48A	Auxiliary function code signals (for group 1 to 4) (PMC axis control)	F132,F142	○	○
	EM11B to EM48B		F135,F145	○	○
	EM11C to EM48C		F138,F148	○	○
	EM11D to EM48D		F141,F151	○	○
	EMBUFA	Buffering disable signal (for group 1 to 4) (PMC axis control)	G142.2	○	○
	EMBUFB		G154.2	○	○
	EMBUFC		G166.2	○	○
	EMBUFD		G178.2	○	○
	EMFA	Auxiliary function strobe signal (for group 1 to 4) (PMC axis control)	F131.0	○	○
	EMFB		F134.0	○	○
	EMFC		F137.0	○	○
	EMFD		F140.0	○	○
	EMF2A	Auxiliary function 2 strobe signal (for group 1 to 4) (PMC axis control)	F131.2	○	○
	EMF2B		F134.2	○	○
	EMF2C		F137.2	○	○
	EMF2D		F140.2	○	○
	EMF3A	Auxiliary function 3 strobe signal (for group 1 to 4) (PMC axis control)	F131.3	○	○
	EMF3B		F134.3	○	○
	EMF3C		F137.3	○	○
	EMF3D		F140.3	○	○
	EMSBKA	Block stop disable signal (for group 1 to 4) (PMC axis control)	G143.7	○	○
	EMSBKB		G155.7	○	○
	EMSBKC		G167.7	○	○
	EMSBKD		G179.7	○	○
	EMZ0 to EMZ15	Extended external machine zero point shift signal	Specifying by parameter No.1280.	○	○
	ENB	Spindle enable signal	F001.4	○	○
	ENB2		F038.2	○	○
	ENB3		F038.3	○	○
	ENB4		F039.1	○	○
	ENBKY	External key input mode selection signal	G066.1	○	○
	EOTNA	Negative-direction overtravel signals (for group 1 to 4) (PMC axis control)	F130.6	○	○
	EOTNB		F133.6	○	○
	EOTNC		F136.6	○	○
	EOTND		F139.6	○	○

Group	Symbol	Signal name	Address	T series	M series
E	EOTPA	Positive-direction overtravel signals (for group 1 to 4) (PMC axis control)	F130.5	○	○
	EOTPB		F133.5	○	○
	EOTPC		F136.5	○	○
	EOTPD		F139.5	○	○
	EOV0	Override 0% signal (PMC axis control)	F129.5	○	○
	EOVC	Override cancellation signal (for group 1 to 4) (PMC axis control)	G150.5	○	○
	EOVCB		G162.5	○	○
	EOVCC		G174.5	○	○
	EOVCD		G186.5	○	○
	EPN0 to EPN13	Extended external workpiece number search signals	G024.0 to G025.5	○	○
	EPNS	External workpiece number search start signal	G025.7	○	○
	EREND	Read completion signal for external data input	F060.0	○	○
	EROV1,EROV2	Rapid traverse override signals (PMC axis control)	G150.0,G150.1	○	○
	ERS	External reset signal	G008.7	○	○
	ERT	Manual rapid traverse selection signal (PMC axis control)	G150.6	○	○
	ERTVA	Automatic screen erasure status in-progress signal	F006.2	○	○
	ESBKA	Block stop signal (for group 1 to 4) (PMC axis control)	G142.3	○	○
	ESBKB		G154.3	○	○
	ESBKC		G166.3	○	○
	ESBKD		G178.3	○	○
	ESCAN	Search cancel signal for external data input	F060.2	○	○
	ESEND	Search completion signal for external data input	F060.1	○	○
	ESKIP	Skip signal (PMC axis control)	X004.6	○	○
	ESOFA	Servo-off signal (for group 1 to 4) (PMC axis control)	G142.4	○	○
	ESOFB		G154.4	○	○
	ESOFC		G166.4	○	○
	ESOFD		G178.4	○	○
	ESRSYC	Spindle command synchronous control signal	G064.6	○	○
	ESSYC1	Spindle command synchronous control signal (for each spindle)	G264.0	○	○
	ESSYC2		G264.1	○	○
	ESSYC3		G264.2	○	○
	ESSYC4		G264.3	○	○
	ESTB	Read signal for external data input	G002.7	○	○
	ESTPA	Axis control temporary stop signal (for group 1 to 4) (PMC axis control)	G142.5	○	○
	ESTPB		G154.5	○	○
	ESTPC		G166.5	○	○
	ESTPD		G178.5	○	○
	ESTPR	Axis immediate stop start signal	G203.3	○	○
	EUI00 to EUI15	Input signals for P-code macro	G082,G083	○	○
	EUO00 to EUO15	Output signals for P-code macro	F084,F085	○	○
	EXCST	Exchange start signal	G536.4	○	○
	EXCED	Exchange completion signal	F536.4	○	○

Group	Symbol	Signal name	Address	T series	M series
E	EXLM, EXLM2, EXLM3	Stored stroke check 1 select signals	G007.6, G531.6, G531.7	O	O
	EXOFA	Motor excitation off state signal (serial spindle)	F047.4	O	O
	EXOFB		F051.4	O	O
	EXOFC		F170.4	O	O
	EXOFD		F268.4	O	O
	EXINP	External input start signal	G058.1	O	O
	EXSTP	External input/output stop signal	G058.2	O	O
	EXOUT	External output start signal	G058.3	O	O
F	F1D	One-digit F code feed signal	G016.7	-	O
	FEED0	Feed zero signal	F066.2	O	O
	FHROV	0.1% step rapid traverse override selection signal	G353.7	O	O
	FIN	End signal	G004.3	O	O
	FCSS1	Cs contour control change completion signal (for each spindle)	F274.0	O	O
	FCSS2		F274.1	O	O
	FCSS3		F274.2	O	O
	FCSS4		F274.3	O	O
	FLANG	Display language switch completion signal	F545.0	O	O
	FRDTEA	Adaptive resonance elimination filter search completion signal (serial spindle)	F307.0	O	O
	FRDTEB		F309.0	O	O
	FRDTEC		F311.0	O	O
	FRDTED		F313.0	O	O
	FRFSMA	Adaptive resonance elimination filter search mode signal (serial spindle)	G304.7	O	O
	FRFSMB		G308.7	O	O
	FRFSMC		G312.7	O	O
	FRFSMD		G316.7	O	O
	FSCSL	Cs contour control change completion signal	F044.1	O	O
	FSPPH	Spindle phase synchronization control completion signal	F044.3	O	O
	FSPPH1	Spindle phase synchronization control completion signal (for each spindle)	F289.0	O	O
	FSPPH2		F289.1	O	O
	FSPPH3		F289.2	O	O
	FSPPH4		F289.3	O	O
	FSPSY	Spindle synchronous speed control completion signal	F044.2	O	O
	FSPSY1	Spindle synchronous speed control completion signal (for each spindle)	F288.0	O	O
	FSPSY2		F288.1	O	O
	FSPSY3		F288.2	O	O
	FSPSY4		F288.3	O	O
	FSYSA	Flexible synchronization control mode selecting signal	F553.4	O	O
	FSYSB		F553.5	O	O
	FSYSC		F553.6	O	O
	FSYSD		F553.7	O	O
	FWSTP	Forward movement prohibition signal	G531.0	O	O

Group	Symbol	Signal name	Address	T series	M series
G	G2RVX	Tool offset direction signal	G090.0	○	-
	G2RVY		G090.2	○	-
	G2RVZ		G090.1	○	-
	G2SLC	2nd geometry tool offset signal	G090.7	○	-
	G2X	2nd geometry tool offset axis select signal	G090.4	○	-
	G2Y		G090.6	○	-
	G2Z		G090.5	○	-
	GAE1	Measuring position reached signals	G517.0	○	○
	GAE2		G517.1	○	○
	GAE3		G517.2	-	○
	GIS	Servo loop gain / in-position width switching signal	G599.3	○	○
	GISO	Servo loop gain / in-position width switching confirmation signal	F599.3	○	○
	GOQSM	Tool offset measurement mode selection signal	G039.7	—	○
	GOQSM	Tool offset write mode select signal	G039.7	○	—
	GQSMC	Tool offset write mode select signal (for milling and turning function)	G546.7	-	○
	GR1,GR2	Gear selection signals (input)	G028.1,G028.2	○	○
	GR1O,GR2O,GR3O	Gear selection signals (output)	F034.0 to F034.2	-	○
	GR21,GR22	Gear selection signals (input)	G029.0,G029.1	○	○
	GR31,GR32		G029.2,G029.3	○	○
	GR41,GR42		G031.4,G031.5	○	○
	GTMC	Groove of thread measurement completion signal	F546.4	○	○
	GTME	Groove of thread measurement error signal	F546.5	○	○
	GTMSR	Groove of thread measurement signal	G549.4	○	○
H	HBTRN	Dual position feedback turning mode selection signal	G531.3	○	○
	HCAB2	Hard copy cancellation request reception signal	F061.2	○	○
	HCABT	Hard copy cancellation request signal	G067.6	○	○
	HCEXE	Hard copy execution status signal	F061.3	○	○
	HCREQ	Hard copy execution request signal	G067.7	○	○
	HCSKP1 to 4	High-speed cycle machining skip selection signals	G065.4 to G065.7	○	○
	HDN	Manual handle feed direction inversion signal	G347.1	○	○
	HDO0 to HDO7	High-speed skip status signals	F122	○	○
	HDSR	Selecting direction of manual handle rotation signal	G193.3	○	○

Group	Symbol	Signal name	Address	T series	M series
H	HEAD	Path select signal (Tool post select signal)	G063.0	●	●
	HEAD2	Path select signal 2 (Tool post select signal 2)	G062.7	●	●
	HEAD3	Path select signal 3 (Tool post select signal 3)	G408.1	●	●
	HEAD4	Path select signal 4 (Tool post select signal 4)	G408.2	●	●
	HF1A	Resonance elimination filter 1 disable signal (serial spindle)	G305.0	○	○
	HF1B		G309.0	○	○
	HF1C		G313.0	○	○
	HF1D		G317.0	○	○
	HF2A	Resonance elimination filter 2 disable signal (serial spindle)	G305.1	○	○
	HF2B		G309.1	○	○
	HF2C		G313.1	○	○
	HF2D		G317.1	○	○
	HF3A	Resonance elimination filter 3 disable signal (serial spindle)	G305.2	○	○
	HF3B		G309.2	○	○
	HF3C		G313.2	○	○
	HF3D		G317.2	○	○
	HF4A	Resonance elimination filter 4 disable signal (serial spindle)	G305.3	○	○
	HF4B		G309.3	○	○
	HF4C		G313.3	○	○
	HF4D		G317.3	○	○
	HNDLF	Manual handle feed maximum feedrate change signal	G023.3	○	○
	HNDMP	Manual pulse magnification change signal	G088.3	○	○
	HPMERR	Programs save error signal	F547.7	○	○
	HPMNTS	Programs not saved status signal	F547.5	○	○
	HPMRSV	All programs save request signal	G514.4	○	○
	HPMSVM	Programs saving in progress signal	F547.6	○	○
	HPS01 to HPS16	High-speed position switch signals	F293,F294	○	○
			Y***,Y***+1	○	○
	HROV	1% step rapid traverse override selection signals	G096.7	○	○
	HREV	Handle-synchronous feed signal	G023.4	○	○
	HS1A to HS1D, HS1E	Manual handle feed axis selection signals	G018.0 to G018.3, G411.0	○	○
	HS1AO	Software operator's panel signal (HS1A)	F077.0	○	○
	HS1BO	Software operator's panel signal (HS1B)	F077.1	○	○
	HS1CO	Software operator's panel signal (HS1C)	F077.2	○	○
	HS1DO	Software operator's panel signal (HS1D)	F077.3	○	○

Group	Symbol	Signal name	Address	T series	M series
H	HS1IA to HS1ID, HS1IE	Manual handle interrupt axis selection signals	G041.0 to G041.3, G411.4	○	○
	HS2A to HS2D, HS2E	Manual handle feed axis selection signals	G018.4 to G018.7, G411.1	○	○
	HS2IA to HS2ID, HS2IE	Manual handle interrupt axis selection signals	G041.4 to G041.7, G411.5	○	○
	HS3A to HS3D, HS3E	Manual handle feed axis selection signals	G019.0 to G019.3, G411.2	○	○
	HS3IA to HS3ID, HS3IE	Manual handle interrupt axis selection signals	G042.0 to G042.3, G411.6	○	○
	HS4A to HS4D, HS4E	Manual handle feed axis selection signals	G020.0 to G020.3, G411.3	○	○
	HS4IA to HS4ID, HS4IE	Manual handle interrupt axis selection signals	G088.4 to G088.7, G411.7	○	○
	HS5A to HS5D, HS5E	Manual handle feed axis selection signals	G379.0 to G379.3, G412.0	○	○
	HS5IA to HS5ID, HS5IE	Manual handle interrupt axis selection signals	G379.4 to G379.7, G412.4	○	○
	HSBIN	High-speed cycle machining in-progress signal	F092.7	○	○
		High-speed machining in-progress signal	F092.7	○	○
	HSRA	High-speed cycle machining retract in-progress signal	F062.2	○	○
		High-speed binary program operation retract in-progress signal	F062.2	○	○
	HSRT	High-speed cycle machining retract selection signal	G065.3	○	○
		High-speed binary program operation retract selection signal	G065.3	○	○
	HSST	High-speed cycle machining immediate stop signal	G589.4	○	○
I	IGNVRY	All-axis VRDY off alarm ignore signal	G066.0	○	○
	IGVRY1 to IGVRY8	Each-axis VRDY off alarm ignore signal	G192	○	○
	INCH	Inch input signal	F002.0	○	○
	INCMDA	Incremental command externally set orientation signals (serial spindle)	G072.5	○	○
	INCMDB		G076.5	○	○
	INCMDC		G206.5	○	○
	INCMDD		G268.5	○	○
	INCSTA	Incremental orientation mode signals (serial spindle)	F047.1	○	○
	INCSTB		F051.1	○	○
	INCSTC		F170.1	○	○
	INCSTD		F268.1	○	○
	INDXA	Orientation stop position change command signals (serial spindle)	G072.0	○	○
	INDXB		G076.0	○	○
	INDXC		G206.0	○	○
	INDXD		G268.0	○	○

Group	Symbol	Signal name	Address	T series	M series
I	INESFNA	Inertia estimation completion signal (serial spindle)	F307.2	○	○
	INESFNB		F309.2	○	○
	INESFNC		F311.2	○	○
	INESFND		F313.2	○	○
	INESTRA	Inertia estimation start signal (serial spindle)	G304.6	○	○
	INESTRB		G308.6	○	○
	INESTRC		G312.6	○	○
	INESTRD		G316.6	○	○
	INFD	In-feed control cut start signal	G063.6	-	○
	INHKY	Key input disable signal	F053.0	○	○
	INIST	Initial axis assignment signal	F536.7	○	○
	INP1 to INP8	In-position signals	F104	○	○
	INTGA	Speed integral signals (serial spindle)	G071.5	○	○
	INTGB		G075.5	○	○
	INTGC		G205.5	○	○
	INTGD		G267.5	○	○
	IOALM	Input/output alarm signal	F053.3	○	○
	IOBSY	Input/output busy signal	F053.2	○	○
	IOLBH1, IOLBH2	Manual handle generators selection signal	G199.0, G199.1	○	○
	IOLBR	β ready signal	F531.7	○	○
	IRTKY	Reset key input invalid signal	G299.7	○	○
	ITF01 to ITF10	Path interference check association signal	G406.0 to G407.1	●	-
	IUDD1 to IUDD8	Unexpected disturbance torque detection ignore signal	G125	○	○
K	KEY1 to KEY4	Memory protection signals	G046.3 to G046.6	○	○
	KEYO	Software operator's panel signal (KEY1 to KEY4)	F075.6	○	○
	KEYP	Memory protection signal	G046.0	○	○
L	LANG1 to LANG7	Display language setting signals	G581.0 to G581.6	○	○
	LDT1A	Load detection signals 1 (serial spindle)	F045.4	○	○
	LDT1B		F049.4	○	○
	LDT1C		F168.4	○	○
	LDT1D		F266.4	○	○
	LDT2A	Load detection signals 2(serial spindle)	F045.5	○	○
	LDT2B		F049.5	○	○
	LDT2C		F168.5	○	○
	LDT2D		F266.5	○	○
	LFCIF	Tool life counting disabled signal	F093.2	○	○
	LFCIV	Tool life counting disable signal	G048.2	○	○
	LIFOVR	Periodic maintenance lifetime warning signal	F093.0	○	○
M	M00 to M31	Auxiliary function code signals	F010 to F013	○	○
	M200 to M215	2nd M function code signals	F014 to F015	○	○
	M200 to M231		F014 to F017		
	M300 to M315	3rd M function code signals	F016 to F017	○	○
	M300 to M331		F564 to F567		
	M400 to M431	4th M function code signals	F568 to F571	○	○
	M500 to M531	5th M function code signals	F572 to F575	○	○

Group	Symbol	Signal name	Address	T series	M series
M	M3R	Three-dimensional coordinate system conversion manual interruption switch signal	G031.3	○	○
	MA	CNC ready signal	F001.7	○	○
	MABSM	Manual absolute check signal	F004.2	○	○
	MAFL	Auxiliary function lock check signal	F004.4	○	○
	MBCAN	Block cancel acknowledgement signal	F297.0	○	○
	MBDT1	Optional block skip check signals	F004.0	○	○
	MBDT2 to MBDT9		F005	○	○
	MBSO	Middle block start signal	F534.4	○	○
	MCEX1 to MCEX16	Call program confirmation signal	F514,F515	○	○
	MCEXE	Macro call executing signal	F512.0	○	○
	MCFIN	Mode change completion signal	G514.0	○	○
	MCFNA	Power line switch completion signals (serial spindle)	G071.3	○	○
	MCFNB		G075.3	○	○
	MCFNC		G205.3	○	○
	MCFND		G267.3	○	○
	MCHK	Handle available signal in checking mode	G067.3	○	○
	MCRQ	Mode change request signal	F512.1	○	○
	MCSP	Abnormal end signal	F512.2	○	○
	MCST1 to MCST16	Macro call start signal	G512,G513	○	○
	MD1,MD2,MD4	Mode selection signals	G043.0 to G043.2	○	○
	MD1O	Software operator's panel signal (MD1)	F073.0	○	○
	MD1R	Mode notification signal	F513.0	○	○
	MD2O	Software operator's panel signal (MD2)	F073.1	○	○
	MD2R	Mode notification signal	F513.1	○	○
	MD4O	Software operator's panel signal (MD4)	F073.2	○	○
	MD4R	Mode notification signal	F513.2	○	○
	MDIRST	MDI reset confirmation signal	F006.1	○	○
	MDOFF1 to MDOFF4	Spindle control mode off signals	G586.4 to G586.7	○	○
	MDRN	Dry run check signal	F002.7	○	○
	MDTCH1 to MDTCH8	Controlled axis detach status signals	F110	○	○
	MEDT	Program edit selection check signal	F003.6	○	○
	MF	Auxiliary function strobe signals	F007.0	○	○
	MF2	2nd M function strobe signal	F008.4	○	○
	MF3	3rd M function strobe signal	F008.5	○	○
	MF4	4th M function strobe signal	F008.6	○	○
	MF5	5th M function strobe signal	F008.7	○	○
	MFIN	Auxiliary function completion signal	G005.0	○	○
	MFIN2	2nd M function completion signal	G004.4	○	○
	MFIN3	3rd M function completion signal	G004.5	○	○

Group	Symbol	Signal name	Address	T series	M series
M	MFIN4	4th M function completion signal	G004.6	○	○
	MFIN5	5th M function completion signal	G004.7	○	○
	MFNHGA	Spindle switch MAIN MCC contact status signals (serial spindle)	G072.6	○	○
	MFNHGB		G076.6	○	○
	MFNHGC		G206.6	○	○
	MFNHGD		G268.6	○	○
	MFSYNA,MFSYNB MFSYNC,MFSYND	Flexible synchronization control mode status signals	F197.0 to F197.3	○	○
	MH	Manual handle feed selection check signal	F003.1	○	○
	MHLC1 to MHLC5	Manual linear/circular interpolation signals	G544.0 to G544.4	○	○
	MHUS1 to MHUS5	Usage selection of manual linear/circular interpolation signals	G545.0 to G545.4	○	○
	MI1 to MI8	Mirror image signals	G106	○	○
	MINC	Incremental feed selection check signal	F003.0	○	○
	MIX1 to MIX8	Composite control axis change selection signals	G128	○	○
	MIXO1 to MIXO8	Composite axis confirmation signals	F343	○	○
	MJ	Jog feed selection check signal	F003.2	○	○
	MLK	All-axis machine lock signal	G044.1	○	○
	MLK1 to MLK8	Each-axis machine lock signal	G108	○	○
	MLKO	Software operator's panel signal (MLK)	F075.4	○	○
	MMDI	Manual data input selection check signal	F003.3	○	○
	MMEM	Automatic operation selection check signal	F003.5	○	○
	MMI1 to MMI8	Mirror image check signals	F108	○	○
	MMLK	All-axis machine lock check signal	F004.1	○	○
	MMMOD	Check mode confirmation signal	F091.3	○	○
	MMOD	Checking mode signal	G067.2	○	○
	MNCHG	Direction change prohibition signal	F091.1	○	○
	MORA1A	Magnetic sensor orientation completion signals (serial spindle)	F046.6	○	○
	MORA1B		F050.6	○	○
	MORA1C		F169.6	○	○
	MORA1D		F267.6	○	○
	MORA2A	Magnetic sensor orientation proximity signals (serial spindle)	F046.7	○	○
	MORA2B		F050.7	○	○
	MORA2C		F169.7	○	○
	MORA2D		F267.7	○	○
	MORCMA	Magnetic sensor orientation command signals (serial spindle)	G073.0	○	○
	MORCMB		G077.0	○	○
	MORCMC		G207.0	○	○
	MORCMD		G269.0	○	○

Group	Symbol	Signal name	Address	T series	M series
M	MP1,MP2,MP4	Manual handle feed amount selection signals (incremental feed signals)	G019.4,G019.5 G019.6	○	○
	MP21,MP22 MP31,MP32 MP41,MP42 MP51,MP52	Manual handle feed amount selection signals	G087.0,G087.1 G087.3,G087.4 G087.6,G087.7 G380.0,G380.1	○	○
	MP1O	Software operator's panel signal (MP1)	F076.0	○	○
	MP2O	Software operator's panel signal (MP2)	F076.1	○	○
	MPOFA	Motor power cutoff command signals (serial spindle)	G073.2	○	○
	MPOFB		G077.2	○	○
	MPOFC		G207.2	○	○
	MPOFD		G269.2	○	○
	MRDYA	Machine ready signals (serial spindle)	G070.7	○	○
	MRDYB		G074.7	○	○
	MRDYC		G204.7	○	○
	MRDYD		G266.7	○	○
	MREF	Manual reference position return selection check signal	F004.5	○	○
	MRMT	DNC operation selection confirm signal	F003.4	○	○
	MRVM	Reverse movement prohibition signal	G531.1	○	○
	MRVMD	Reverse movement signal	F091.0	○	○
	MRVSP	Reverse movement prohibition signal	F091.2	○	○
	MSBK	Single block check signal	F004.3	○	○
	MSEMI1 to MSEMI8	Position feedback check signals	F516	○	○
	MSP00 to MSP15	Multi-spindle address P signals	F160,F161	○	○
	MSPOSA	Spindle positioning mode signals	F039.0	○	○
	MSPOSB		F402.1	○	○
	MSPOSC		F402.2	○	○
	MSPOSD		F402.3	○	○
	MT8N00 to MT8N31	Manual tool compensation tool number signal (8 digits)	G525 to G528	○	-
	MTA,MTB,MTC, MTD	Flexible synchronization control mode selection signals	G197.0 to G197.3	○	○
	MTLA	Manual tool compensation completion signal	F061.5	○	-
	MTLANG	Manual tool compensation uncompleted signal	F061.4	○	-
	MTLC	Manual tool compensation command number	G067.0	○	-
	MTLN00 to MTLN15	Manual tool compensation tool number signal (4 digits)	G068,G069	○	-
	MV1 to MV8	Axis moving signals	F102	○	○
	MVD1 to MVD8	Axis moving direction signals	F106	○	○
N	NDCAL1 to NDCAL8	A/B phase detector disconnection alarm ignore signal (PMC axis control)	G202	○	○
	NHSW	Waiting M codes of high-speed type invalid signal	G579.6	●	●
		Speed-up of non-buffering command by G code invalid signal		○	○
	NMWT	No-wait signal	G063.7	●	●
	NOINPS	In-position check disable signal	G023.5	○	○
	NOT3DM	3-dimensional coordinate system conversion manual interrupt enable/disable switch signal	G347.7	○	○

Group	Symbol	Signal name	Address	T series	M series
N	NOWT	No-wait signal	G063.1	●	●
	NOZAGC	Signal for disabling angular axis control for the perpendicular axis	G063.5	○	○
	NPOS1 to NPOS8	Axis non-displayed signals	G198	○	○
	NRROA	Shortcut command signals for orientation stop position change (serial spindle)	G072.2	○	○
	NRROB		G076.2	○	○
	NRROC		G206.2	○	○
	NRROD		G268.2	○	○
	NSYNCA	Signal for disabling torque difference alarm detection for axis synchronous control	G059.7	○	○
O	OFN0 to OFN5, OFN6 to OFN9	Tool offset number selection signals	G039.0 to G039.5, G040.0 to G040.3	○	○
	OFNC0 to OFNC5, OFNC6 to OFNC9	Tool offset number selection signals (for milling and turning function)	G546.0 to G546.5, G547.0 to G547.3	-	○
	ONSC	Tool compensation number specification signal	G547.6	○	–
	OP	Automatic operation signal	F000.7	○	○
	ORARA	Orientation completion signals (serial spindle)	F045.7	○	○
	ORARB		F049.7	○	○
	ORARC		F168.7	○	○
	ORARD		F266.7	○	○
	ORCMA	Orientation command signals(serial spindle)	G070.6	○	○
	ORCMB		G074.6	○	○
	ORCMC		G204.6	○	○
	ORCMD		G266.6	○	○
	OTA1 to OTA8	Stored stroke limit range switching axis selection signals	G596	○	○
	OTD0 to OTD15	Stored stroke limit range switching data selection signals	G594, G595	○	○
	OTSW	Stored stroke limit range switching start signal	G599.0	○	○
	OTSWFN	Stored stroke limit range switching finish signal	F599.0	○	○
	OUT0 to OUT15	Software operator's panel general-purpose switch signals	F072,F074	○	○
	OVC	Override cancel signal	G006.4	○	○
	OVLN	Inter-path flexible synchronization mode select signal Advanced superimposition signal AI contour control permission signal	G531.4	○	○
	OVLNS	Inter-path flexible synchronization mode signal Advanced superimposition mode signal	F545.1	○	○
	OVLS1 to OVLS8	Superimposed control axis selection signals	G190	○	○
	OVMO1 to OVMO8	Superimposed control master axis confirmation signals	F344	○	○
	OVRA	Analog override signals (serial spindle)	G072.4	○	○
	OVRB		G076.4	○	○
	OVRC		G206.4	○	○
	OVRD		G268.4	○	○
	OVSO1 to OVSO8	Superimposed control slave axis confirmation signals	F345	○	○

Group	Symbol	Signal name	Address	T series	M series
P	P1SI1 to P1SI16	Pulse superimposed command signals	G574, G575	○	○
	P2SI1 to P2SI16		G576, G577	○	○
	PBATL	Absolute position detector battery voltage low alarm signal	F172.7	○	○
	PBATZ	Absolute position detector battery voltage zero alarm signal	F172.6	○	○
	PC1DTA	Position coder one-rotation signal detection status signals (serial spindle)	F047.0	○	○
	PC1DTB		F051.0	○	○
	PC1DTC		F170.0	○	○
	PC1DTD		F268.0	○	○
	PC2SLC	2nd position coder selection signal	G028.7	○	○
	PC3SLC	3rd position coder selection signal	G026.0	○	○
	PC4SLC	4th position coder selection signal	G026.1	○	○
	PCKSV	High speed program check saving data signal	F290.4	○	○
	PECK2	Small-hole peck drilling cycle in progress signal	F066.5	-	○
	PGCK	High-speed program check signal	G290.5	○	○
	PHERA, PHERB, PHERC, PHERD	Automatic phase synchronization error detection signals	F553.0 to F553.3	○	○
	PHFINA to PHFIND	Flexible synchronization control phase synchronization end signals	F381.0 to F381.3	○	○
	PK1 to PK8	Parking signals	G122	○	○
	PKESS1	1st spindle parking signal	G122.6 (G031.6)	○	○
	PKESS2	2nd spindle parking signal	G122.7 (G031.7)	○	○
	PKESE1	Spindle command synchronous parking signal(for each spindle)	G265.0	○	○
	PKESE2		G265.1	○	○
	PKESE3		G265.2	○	○
	PKESE4		G265.3	○	○
	PN1,PN2,PN4, PN8,PN16	External workpiece number search signals	G009.0 to G009.4	○	○
	PORA2A	Position coder orientation proximity signal (serial spindle)	F046.5	○	○
	PORA2B		F050.5	○	○
	PORA2C		F169.5	○	○
	PORA2D		F267.5	○	○
	PRC	Position record signal	G040.6	○	-
	PRGDPL	Program screen display mode signal	F053.1	○	○
	PRGMD	High speed program check mode signal	F290.5	○	○
	PRTSF	Target part count reached signal	F062.7	○	○
	PSAR	Polygon spindle speed arrival signal	F063.2	○	○
	PSE1	Polygon master axis not arrival signal	F063.0	○	○
	PSE2	Polygon synchronization axis not arrival signal	F063.1	○	○
	PSI1	Pulse superimposed strobe signals	G578.0	○	○
	PSI2		G578.1	○	○
	PSIF1	Pulse superimposed command reading completion signals	F544.0	○	○
	PSIF2		F544.1	○	○
	PSIM	Pulse superimposed mode switching signal	G578.7	○	○
	PSIMF	Pulse superimposed mode signal	F544.7	○	○
	PSW01 to PSW16	Position switch signals	F070,F071	○	○
	PSYN	Polygon synchronization under way signal	F063.7	○	○
	PWFL	Power failure deceleration signal	G203.7	○	○

Group	Symbol	Signal name	Address	T series	M series
P	PWMSEA	Cutting feed/rapid traverse PWM frequency switching function in Cs contour control enable signal (serial spindle)	G306.1	○	○
	PWMSEB		G310.1	○	○
	PWMSEC		G314.1	○	○
	PWMSED		G318.1	○	○
Q	QRSTD	Program restart memory storing disabled signal	G517.6	○	○
R	R01I to R12I	Spindle motor speed command signals	G032.0 to G033.3	○	○
	R01I2 to R12I2		G034.0 to G035.3	○	○
	R01I3 to R12I3		G036.0 to G037.3	○	○
	R01I4 to R12I4		G272.0 to G273.3	○	○
	R01O to R12O	S 12-bit code signals	F036.0 to F037.3	○	○
	R01O2 to R12O2		F200.0 to F201.3	○	○
	R01O3 to R12O3		F204.0 to F205.3	○	○
	R01O4 to R12O4		F270.0 to F271.3	○	○
	RCFNA	Output switch completion signals (serial spindle)	F046.3	○	○
	RCFNB		F050.3	○	○
	RCFNC		F169.3	○	○
	RCFND		F267.3	○	○
	RCHA	Power line status check signals (serial spindle)	G071.7	○	○
	RCHB		G075.7	○	○
	RCHC		G205.7	○	○
	RCHD		G267.7	○	○
	RCHHGA	Spindle switch HIGH MCC contact status signals (serial spindle)	G072.7	○	○
	RCHHGB		G076.7	○	○
	RCHHGC		G206.7	○	○
	RCHHGD		G268.7	○	○
	RCHPA	Output switch signals (serial spindle)	F046.2	○	○
	RCHPB		F050.2	○	○
	RCHPC		F169.2	○	○
	RCHPD		F267.2	○	○
	RE01I to RE32I	Extended spindle motor speed command signals	G708 to G711	○	○
	RE01I2 to RE32I2		G712 to G715	○	○
	RE01I3 to RE32I3		G716 to G719	○	○
	RE01I4 to RE32I4		G720 to G723	○	○
	RE01O to RE32O	S32-bit code signals	F708 to F711	○	○
	RE01O2 to RE32O2		F712 to F715	○	○
	RE01O3 to RE32O3		F716 to F719	○	○
	RE01O4 to RE32O4		F720 to F723	○	○
	RGHTH	Tool axis right-angle direction feed mode signal	G023.6	○	○
	RGSPM	Spindle rotation direction signals	F065.1	○	○
	RGSPP		F065.0	○	○
	RGTAP	Rigid tapping signal	G061.0	○	○
	RGTSP1 to RGTSP4	Rigid tapping spindle selection signals	G061.4 to G061.7	○	-
	RLSOT	Stroke check 1 release signal	G007.7	○	○
	RLSOT3	Stroke check 3 release signal	G007.4	○	○
	RMTC	Re-machining thread signal	G549.5	○	○
	RMVST	Removal start signal	G536.2	○	○
	RMVED	Removal completion signal	F536.2	○	○
	RNDH	Tool tip center rotation feed mode signal	G298.2	○	○

Group	Symbol	Signal name	Address	T series	M series
R	ROTAA	Rotational direction command signals for orientation stop position change (serial spindle)	G072.1	○	○
	ROTAB		G076.1	○	○
	ROTAC		G206.1	○	○
	ROTAD		G268.1	○	○
	ROV1,ROV2	Rapid traverse override signals	G014.0,G014.1	○	○
	ROV1O	Software operator's panel signal (ROV1)	F076.4	○	○
	ROV2O	Software operator's panel signal (ROV2)	F076.5	○	○
	ROVLP	Rapid traverse block overlap disable signal	G053.5	○	○
	RP11 to RP18	Reference position match signals	F517.0 to F517.7	○	○
	RP21 to RP28	2nd reference position match signals	F518.0 to F518.7	○	○
	RPDO	Rapid traversing signal	F002.1	○	○
	RRW	Reset & rewind signal	G008.6	○	○
	RSLA	Output switch request signals (serial spindle)	G071.6	○	○
	RSLB		G075.6	○	○
	RSLC		G205.6	○	○
	RSLD		G267.6	○	○
	RSMAX	Spindle synchronous speed ratio control clamp signal	F065.2	○	○
	RST	Resetting signal	F001.1	○	○
	RT	Manual rapid traverse selection signal	G019.7	○	○
	RTAP	Rigid tapping-in-progress signal	F076.3	○	○
	RTC2	Time constant of acceleration / deceleration after interpolation for rapid traverse switching signals	G599.6	○	○
	RTC3		G599.7	○	○
	RTC2O	Time constant of acceleration / deceleration after interpolation for rapid traverse switching confirmation signals	F599.6	○	○
	RTC3O		F599.7	○	○
	RTNT	Rigid tapping retraction start signal	G062.6	○	○
	RTO	Software operator's panel signal (RT)	F077.6	○	○
	RTPT	Rigid tapping retraction completion signal	F066.1	○	○
	RTRCT	Retract signal	G066.4	○	○
	RTRCTF	Retract completion signal	F065.4	○	○
	RVS	Reverse execution signal	G007.0	-	○
	RWD	Rewinding signal	F000.0	○	○
S	S00 to S31	Spindle function code signals	F022 to F025	○	○
	S1MES	Spindle 1 under measurement signal	F062.3	○	-
	S2MES	Spindle 2 under measurement signal	F062.4	○	-
	S2TLS	Spindle measurement select signal	G040.5	○	-
	SA	Servo ready signal	F000.6	○	○
	SAR	Spindle speed arrival signal	G029.4	○	○
	SARA	Spindle speed arrival signal (serial spindle)	F045.3	○	○
	SARB		F049.3	○	○
	SARC		F168.3	○	○
	SARD		F266.3	○	○
	SBK	Single block signal	G046.1	○	○
	SBKO	Software operator's panel signal (SBK)	F075.3	○	○
	SBRT	Spindle synchronous speed ratio control signal	G038.1	○	○

Group	Symbol	Signal name	Address	T series	M series
S	SCLPA	Spindle clamp signal	F038.0	○	○
	SCLPB		F401.1	○	○
	SCLPC		F401.2	○	○
	SCLPD		F401.3	○	○
	SDPC	Speed display change signal	G038.5	○	○
	SDTA	Speed detection signals (serial spindle)	F045.2	○	○
	SDTB		F049.2	○	○
	SDTC		F168.2	○	○
	SDTD		F266.2	○	○
	SEO1 to SEO8	Excess synchronization error signals	F559	○	○
	SEMI1 to SEMI8	Position feedback dynamic switching signals	G516	○	○
	SF	Spindle function strobe signal	F007.2	○	○
	SFAN	Alarm level detection signal	F093.1	○	○
	SFIN	Spindle function completion signal	G005.2	○	○
	SFRA	CW command signals (serial spindle)	G070.5	○	○
	SFRB		G074.5	○	○
	SFRC		G204.5	○	○
	SFRD		G266.5	○	○
	SGN	Spindle motor command polarity command signals	G033.5	○	○
	SGN2		G035.5	○	○
	SGN3		G037.5	○	○
	SGN4		G273.5	○	○
	SH00A to SH14A	Spindle orientation signals with the stop position externally set	G078.0 to G079.6	○	○
	SH00B to SH14B		G080.0 to G081.6	○	○
	SH00C to SH14C		G208.0 to G209.6	○	○
	SH00D to SH14D		G270.0 to G271.6	○	○
	SIND	Spindle motor speed command selection signals	G033.7	○	○
	SIND2		G035.7	○	○
	SIND3		G037.7	○	○
	SIND4		G273.7	○	○
	SKIP	Skip signal	X004.7	○	○
	SKIP2 to SKIP6, SKIP7,SKIP8	Skip signal	X004.2 to X004.6, X004.0,X004.1	○	○
	SKIPP	Skip signal	G006.6	○	○
	SLANG	Display language switch start signal	G581.7	○	○
	SLPCA	Path spindle feedback selection signals	G064.2	●	●
	SLPCB		G064.3	●	●
	SLPCC		G403.4	●	●
	SLPCD		G403.5	●	●
	SLREF	Manual 2nd/3rd/4th reference position return select 1 signal	G340.5	○	○
	SLRER	Manual 2nd/3rd/4th reference position return select 2 signal	G340.6	○	○
	SLSPA	Path spindle command selection signals	G063.2	●	●
	SLSPB		G063.3	●	●
	SLSPC		G403.0	●	●
	SLSPD		G403.1	●	●

Group	Symbol	Signal name	Address	T series	M series
S	SLVA	Subordinate operation mode command signals (serial spindle)	G073.1	○	○
	SLVB		G077.1	○	○
	SLVC		G207.1	○	○
	SLVD		G269.1	○	○
	SLVSA	Subordinate operation status signals (serial spindle)	F046.4	○	○
	SLVSB		F050.4	○	○
	SLVSC		F169.4	○	○
	SLVSD		F267.4	○	○
	SMPK1 to SMPK8	Parking axis confirmation signals	F346	○	○
	SMSL11	Spindle position save selection signal	G588.0	○	○
	SMSL12		G588.1	○	○
	SMSL13		G588.2	○	○
	SMSL14		G588.3	○	○
	SMSL21		G588.4	○	○
	SMSL22		G588.5	○	○
	SMSL23		G588.6	○	○
	SMSL24		G588.7	○	○
	SMZ	In-position check signal	G053.6	○	○
	SOCNA	Soft start/stop signals (serial spindle)	G071.4	○	○
	SOCNB		G075.4	○	○
	SOCNC		G205.4	○	○
	SOCND		G267.4	○	○
	SOR	Spindle orientation signal	G029.5	○	○
	SORENA	Synchronous orientation enable signal	F047.3	○	○
	SORENB		F051.3	○	○
	SORENC		F170.3	○	○
	SOREND		F268.3	○	○
	SORSLA	Synchronous orientation request command	G073.3	○	○
	SORSLB		G077.3	○	○
	SORSLC		G207.3	○	○
	SORSLD		G269.3	○	○
	SOV0 to SOV7	Spindle speed override signals	G030	○	○
	SOV20 to SOV27	2nd spindle speed override signals	G376	○	○
	SOV30 to SOV37	3rd spindle speed override signals	G377	○	○
	SOV40 to SOV47	4th spindle speed override signals	G378	○	○
	SPAL	Spindle speed fluctuation detection alarm signal	F035.0	○	○
	SPAPH1	Arbitrary spindle position phase synchronization signal	G587.4	○	○
	SPAPH2		G587.5	○	○
	SPAPH3		G587.6	○	○
	SPAPH4		G587.7	○	○
	SPL	Feed hold lamp signal	F000.4	○	○
	SPMER1	Spindle position save error signal	F577.4	○	○
	SPMER2		F577.5	○	○
	SPMER3		F577.6	○	○
	SPMER4		F577.7	○	○
	SPMFN1	Spindle position save completion signal	F577.0	○	○
	SPMFN2		F577.1	○	○
	SPMFN3		F577.2	○	○
	SPMFN4		F577.3	○	○

Group	Symbol	Signal name	Address	T series	M series
S	SPMST1	Spindle position save start signal	G587.0	○	○
	SPMST2		G587.1	○	○
	SPMST3		G587.2	○	○
	SPMST4		G587.3	○	○
	SPO	Software operator's panel signal (*SP)	F075.7	○	○
	SPP1 to SPP8	Spindle indexing signals for each axis	F522	○	○
	SPPHS	Spindle phase synchronous control signal	G038.3	○	○
	SPPHS1	Spindle phase synchronous control signal (for each spindle)	G289.0	○	○
	SPPHS2		G289.1	○	○
	SPPHS3		G289.2	○	○
	SPPHS4		G289.3	○	○
	SPSLA	Spindle selection signals (serial spindle)	G071.2	○	○
	SPSLB		G075.2	○	○
	SPSLC		G205.2	○	○
	SPSLD		G267.2	○	○
	SPSP	Spindle command path specification signal	G536.7	○	○
	SPSTPA	Spindle stop complete signal	G028.6	○	○
	SPSTPB		G402.1	○	○
	SPSTPC		G402.2	○	○
	SPSTPD		G402.3	○	○
	SPSYC	Spindle synchronous control signal	G038.2	○	○
	SPSYC1	Spindle synchronous control signal (for each spindle)	G288.0	○	○
	SPSYC2		G288.1	○	○
	SPSYC3		G288.2	○	○
	SPSYC4		G288.3	○	○
	SPWRN1 to SPWRN9	Spindle warning detail signals 1 to 9	F264.0 to F265.0	○	○
	SQMPE	Program restart MDI program execution completion signal	F316.7	○	○
	SQMPR	Program restart MDI program output completion signal	F316.6	○	○
	SRN	Program restart signal	G006.0	○	○
	SRNEX	Quick program restart under way signal	Fn534.1	○	○
	SRNMV	Program restart under way signal	F002.4	○	○
	SRSP1R	1st serial spindle ready signals	F034.6	○	○
	SRSP2R	2nd serial spindle ready signals	F034.5	○	○
	SRSP3R	3rd serial spindle ready signals	F034.4	○	○
	SRSP4R	4th serial spindle ready signals	F034.3	○	○
	SRSRDY	All-spindle operation ready signal	F034.7	○	○
	SRVA	CCW command signals (serial spindle)	G070.4	○	○
	SRVB		G074.4	○	○
	SRVC		G204.4	○	○
	SRVD		G266.4	○	○
	SRVON1 to SRVON8	SV speed control mode signals	G521	○	○
	SSEGB1 to SSEGB4	Simple spindle EGB signals	G351.0 to G351.3	○	○
	SSEGBM1 to SSEGBM4	Simple spindle EGB mode signals	F351.0 to F351.3	○	○

Group	Symbol	Signal name	Address	T series	M series
S	SSIN	Spindle motor command polarity selection signals	G033.6	○	○
	SSIN2		G035.6	○	○
	SSIN3		G037.6	○	○
	SSIN4		G273.6	○	○
	SSR1 to SSR4	Total spindle revolution number reset signals	G533.0 to G533.3	○	○
	SSRS	Total spindle revolution number reset selection signal	G533.4	○	○
	SSTA	Speed zero signals (serial spindle)	F045.1	○	○
	SSTB		F049.1	○	○
	SSTC		F168.1	○	○
	SSTD		F266.1	○	○
	ST	Cycle start signal	G007.2	○	○
	STCHK	Start check signal	G408.0	○	○
	STL	Cycle start lamp signal	F000.5	○	○
	STLK	Start lock signal	G007.1	○	○
	SUCLPA	Spindle unclamp signal	F038.1	○	○
	SUCLPB		F400.1	○	○
	SUCLPC		F400.2	○	○
	SUCLPD		F400.3	○	○
	SVAR	Servo motor spindle control mode acceleration/deceleration completion signal	F090.6	○	○
	SVF1 to SVF8	Servo off signals	G126	○	○
	SVGN	Servo motor rotation polarity specification signal	G022.5	○	○
	SVMWC1 to SVMWC8	Dual control axes switching signal	G687	○	○
	SVMWS1 to SVMWS8	Dual control axes status signal	F687	○	○
	SVR01I to SVR12I	Servo motor rotation speed specification signals	G021.0 to G022.3	○	○
	SVREV1 to SVREV8	SV speed control mode in-progress signals	F521	○	○
	SVRVS1 to SVRVS8	SV reverse signals	G523	○	○
	SVSAR1 to SVSAR8	Speed arrival signals	F377	○	○
	SVSP	Servo motor spindle control switching signal	G022.7	○	○
	SVSPM	Servo motor spindle control mode signal	F090.7	○	○
	SVSST1 to SVSST8	Speed zero signals	F376	○	○
	SVWRN1	Servo warning detail signals	F093.4	○	○
	SVWRN2		F093.5	○	○
	SVWRN3		F093.6	○	○
	SVWRN4		F093.7	○	○
	SWS1	Spindle selection signals	G027.0	○	○
	SWS2		G027.1	○	○
	SWS3		G027.2	○	○
	SWS4		G026.3	○	○
	SYAR	Servo motor spindle synchronization mode acceleration/deceleration completion signal	F090.4	○	○
	SYCAL	Phase error monitor signal	F044.4	○	○
	SYCAL1	Phase error monitor signal (for each spindle)	F043.0	○	○
	SYCAL2		F043.1	○	○
	SYCAL3		F043.2	○	○
	SYCAL4		F043.3	○	○

Group	Symbol	Signal name	Address	T series	M series
S	SYCM1 to SYCM8	Synchronous master axis confirmation signals	F341	○	○
	SYCS1 to SYCS8	Synchronous slave axis confirmation signals	F342	○	○
	SYN1O to SYN8O	Synchronous/composite/superimposed control under way signals	F118	○	○
	SYNC1 to SYNC8	Synchronous control axis selection signals	G138	○	○
	SYNCJ1 to SYNCJ8	Signals for selecting the manual feed axis for axis synchronous control	G140	○	○
	SYNER	Signal for indicating a positional deviation error alarm for axis synchronous control	F403.0	○	○
	SYNMOD	EGB mode signal	F065.6	○	○
	SYNMT1 to SYNMT8	Machine coordinate match state output signals	F210	○	○
	SYNO1 to SYNO8	Axis synchronous control status signals	F532	○	○
	SYNOF1 to SYNOF8	Synchronization compensation enable state output signals	F211	○	○
	SYPER	Phase synchronization for Servo/Spindle synchronous error signal	F527.7	○	○
	SYPFN	Phase synchronization for Servo/Spindle synchronous finished signal	F527.6	○	○
	SYPST	Phase synchronization for Servo/Spindle synchronous start signal	G517.7	○	○
	SYSS	Servo motor spindle synchronization start signal	G61.2	○	○
	SYSSM	Servo motor spindle synchronization mode signal	F090.5	○	○
T	T00 to T31	Tool function code signals	F026 to F029	○	○
	TAP	Tapping signal	F001.5	○	○
	TB_BASE	Table base signal	G298.0	○	○
	TDA1 to TDA8	Target distance attainment status signal	F730	○	○
	TDC1 to TDC8	Total travel distance clear signal	G726	○	○
	TDCF00 to TDCF07	DO signal for Data transfer between PMC and DCSPMC	F747	○	○
	TDFCANA	Preload and multi-axis integrator copy disable signal (serial spindle)	G306.2	○	○
	TDFCANB		G310.2	○	○
	TDFCANC		G314.2	○	○
	TDFCAND		G318.2	○	○
	TDFTR1 to TDFTR8	Trouble forecast signals for disturbance level	F299	○	○
	TDSML1 to TDSML8	Trouble forecast signals for thermal simulation	F298	○	○
	TF	Tool function strobe signal	F007.3	○	○
	TFIN	Tool function completion signal	G005.3	○	○
	THRD	Threading signal	F002.3	○	○
	TIALM	Path interference alarm signal	F064.7	●	-
	TICHK	Path interference check in progress signal	F064.6	●	-
	TKEY0 to TKEY5	Tool management data protection signal	G330.0 to G330.5	○	○
	TL01 to TL512	Tool group number selection signals	G047.0 to G048.1	○	○
	TLAL	Number of remaining tools notification signal	F154.0	-	○
	TLCH	Tool change signal	F064.0	○	○
	TLCH1	Tool change signal 1	F328.0	○	○
	TLCH2	Tool change signal 2	F328.1	○	○

Group	Symbol	Signal name	Address	T series	M series
T	TLCH3	Tool change signal 3	F328.2	○	○
	TLCH4	Tool change signal 4	F328.3	○	○
	TLCHB	Tool life expiration notice signal	F064.3	○	○
	TLCHB1	Tool life expiration notice signal 1	F329.4	○	○
	TLCHB2	Tool life expiration notice signal 2	F329.5	○	○
	TLCHB3	Tool life expiration notice signal 3	F329.6	○	○
	TLCHB4	Tool life expiration notice signal 4	F329.7	○	○
	TLCHI	Individual tool change signal	F064.2	○	○
	TLCHI1	Individual tool change signal 1	F328.4	○	○
	TLCHI2	Individual tool change signal 2	F328.5	○	○
	TLCHI3	Individual tool change signal 3	F328.6	○	○
	TLCHI4	Individual tool change signal 4	F328.7	○	○
	TLMA	Torque limit state signals (serial spindle)	F045.6	○	○
	TLMB		F049.6	○	○
	TLMC		F168.6	○	○
	TLMD		F266.6	○	○
	TLMEM	Tool management data edit in-progress signal	F315.7	○	○
	TLMG10	Tool management data modification in-progress signal	F315.2	○	○
	TLMHA	Torque limit command HIGH signals (serial spindle)	G070.1	○	○
	TLMHB		G074.1	○	○
	TLMHC		G204.1	○	○
	TLMHD		G266.1	○	○
	TLMLA	Torque limit command LOW signals (serial spindle)	G070.0	○	○
	TLMLB		G074.0	○	○
	TLMLC		G204.0	○	○
	TLMLD		G266.0	○	○
	TLMOT	Tool management data output in-progress signal	F315.4	○	○
	TLMSRH	Tool search in-progress signal	F315.1	○	○
	TLNCT1	Tool life counting disable signal 1	G329.4	○	○
	TLNCT2	Tool life counting disable signal 2	G329.5	○	○
	TLNCT3	Tool life counting disable signal 3	G329.6	○	○
	TLNCT4	Tool life counting disable signal 4	G329.7	○	○
	TLNW	New tool select signal	F064.1	○	○
	TLRST	Tool change reset signal	G048.7	○	○
	TLRST1	Tool change reset signal 1	G328.0	○	○
	TLRST2	Tool change reset signal 2	G328.1	○	○
	TLRST3	Tool change reset signal 3	G328.2	○	○
	TLRST4	Tool change reset signal 4	G328.3	○	○
	TLRSTI	Individual tool change reset signal	G048.6	○	○
	TLRSTI1	Individual tool change reset signal 1	G328.4	○	○
	TLRSTI2	Individual tool change reset signal 2	G328.5	○	○
	TLRSTI3	Individual tool change reset signal 3	G328.6	○	○
	TLRSTI4	Individual tool change reset signal 4	G328.7	○	○
	TLSKF	Tool skip completion signal	F315.0	○	○
	TLSKF1	Tool skip completion signal 1	F329.0	○	○
	TLSKF2	Tool skip completion signal 2	F329.1	○	○
	TLSKF3	Tool skip completion signal 3	F329.2	○	○
	TLSKF4	Tool skip completion signal 4	F329.3	○	○
	TLSKP	Tool skip signal	G048.5	○	○

Group	Symbol	Signal name	Address	T series	M series
T	TLSKP1	Tool skip signal 1	G329.0	○	○
	TLSKP2	Tool skip signal 2	G329.1	○	○
	TLSKP3	Tool skip signal 3	G329.2	○	○
	TLSKP4	Tool skip signal 4	G329.3	○	○
	TMFNFD	Life expiration signal	F315.6	○	○
	TMRON	General-purpose integrating meter start signal	G053.0	○	○
	TPMG00 to TPMG07	DI signal for Data transfer between PMC and DCSPMC	G765	○	○
	TPPRS	Touch panel check signal	F006.0	○	○
	TRACT	Tool retraction mode signal	F092.3	○	○
	TRESC	Tool retraction signal	G059.0	○	○
	TRMTN	Tool retraction axis movement signal	F092.4	○	○
	TRQL1 to TRQL8	Torque limit reached signals	F114	○	○
	TRQM1 to TRQM8	Torque control mode signal (PMC axis control)	F190	○	○
	TRRTN	Tool return signal	G059.1	○	○
	TRSPS	Tool return completion signal	F092.5	○	○
U	UI000 to UI031	Custom macro input signals	G054 to G057	○	○
	UI100 to UI131		G276 to G279	○	○
	UI200 to UI231		G280 to G283	○	○
	UI300 to UI331		G284 to G287	○	○
	UINT	Interrupt signal for custom macro	G053.3	○	○
	UO000 to UO031	Custom macro output signals	F054,F055, F276,F277	○	○
	UO100 to UO131		F056 to F059	○	○
	UO200 to UO231		F280 to F283	○	○
	UO300 to UO331		F284 to F287	○	○
V	VDCABA	DC-link failure detection state signal (serial spindle)	F306.4	○	○
	VDCABB		F308.4	○	○
	VDCABC		F310.4	○	○
	VDCABD		F312.4	○	○
W	WATO	Waiting signal	F063.6	●	●
	WBCNT	Web browser connection status signal	F0578.2	○	○
	WBEND	Web browser connection prohibition signal	G0579.5	○	○
	WECCS	SRAM ECC abnormality warning signal	F0535.3	○	○
	WETE	Embedded Ethernet communication abnormality warning signal	F0535.4	○	○
	WETF	Fast Ethernet communication abnormality warning signal	F0535.5	○	○
	WFAN	Warning level detection signal	F093.3	○	○
	WFLN1	FL-net1 communication abnormality warning signal	F0535.6	○	○
	WFLN2	FL-net2 communication abnormality warning signal	F0535.7	○	○
	WIOCH1	I/O Link 1 retry abnormality warning signal	F0535.0	○	○
	WIOCH2	I/O Link 2 retry abnormality warning signal	F0535.1	○	○
	WIOCH3	I/O Link 3 retry abnormality warning signal	F0535.2	○	○
	WOQSM	Workpiece origin offset measurement mode selection signal	G039.6	–	○
	WOQSM	Workpiece coordinate system shift value write mode select signal	G039.6	○	–
	WOSET	Workpiece coordinate system shift value write signal	G040.7	○	-

Group	Symbol	Signal name	Address	T series	M series
W	WPRST1 to WPRST8	Each axis workpiece coordinate system preset signals	G358	○	○
	WPSF1 to WPSF8	Each axis workpiece coordinate system preset completion signals	F358	○	○
X	XPFLA	Power failure detection signal (serial spindle)	F307.1	○	○
	XPFLB		F309.1	○	○
	XPFLC		F311.1	○	○
	XPFLD		F313.1	○	○
Z	ZP1 to ZP8	Reference position return end signals	F094	○	○
	ZP21 to ZP28	2nd reference position return completion signals	F096	○	○
	ZP31 to ZP38	3rd reference position return completion signals	F098	○	○
	ZP41 to ZP48	4th reference position return completion signals	F100	○	○
	ZRF1 to ZRF8	Reference position establishment signals	F120	○	○
	ZRN	Manual reference position return selection signal	G043.7	○	○
	ZRNO	Software operator's panel signal (ZRN)	F073.4	○	○
	ZRNR	Mode notification signal	F513.7	○	○

A.2.3 List of Signals (In Order of Addresses)

○ : Available
● : Available only with multi path control
- : Unavailable

Address	Signal name	Symbol	T series	M series
X004.2 to X004.6, X004.0,X004.1	Skip signal	SKIP2 to SKIP6, SKIP7,SKIP8	○	○
X004.2 to X004.5	Tool offset write signals	+MIT1,-MIT1 +MIT2,-MIT2	○	-
X004.6	Skip signal (PMC axis control)	ESKIP	○	○
X004.7	Skip signal	SKIP	○	○
X008.0	Emergency stop signals	*ESP	○	○
X008.1			○	○
X008.4			○	○
X009	Reference position return deceleration signals	*DEC1 to *DEC8	○	○
Y***	High-speed position switch signals	HPS01 to HPS16	○	○
Y***+1				
G000 to G001	Data signals for external data input	ED15 to ED0	○	○
G002.6 to G002.0	Address signals for external data input	EA6 to EA0	○	○
G002.7	Read signal for external data input	ESTB	○	○
G004.3	End signal	FIN	○	○
G004.4	2nd M function completion signal	MFIN2	○	○
G004.5	3rd M function completion signal	MFIN3	○	○
G004.6	4th M function completion signal	MFIN4	○	○
G004.7	5th M function completion signal	MFIN5	○	○
G005.0	Auxiliary function completion signal	MFIN	○	○
G005.2	Spindle function completion signal	SFIN	○	○
G005.3	Tool function completion signal	TFIN	○	○
G005.6	Auxiliary function lock signal	AFL	○	○
G005.7	2nd auxiliary function completion signal	BFIN	○	○
G006.0	Program restart signal	SRN	○	○
G006.2	Manual absolute signal	*ABSM	○	○
G006.4	Override cancel signal	OVC	○	○
G006.6	Skip signal	SKIPP	○	○
G007.0	Reverse execution signal	RVS	-	○
G007.1	Start lock signal	STLK	○	○
G007.2	Cycle start signal	ST	○	○
G007.4	Stroke check 3 release signal	RLSOT3	○	○
G007.5	Follow-up signal	*FLWU	○	○
G007.6	Stored stroke check 1 select signals	EXLM	○	○
G007.7	Stroke check 1 release signal	RLSOT	○	○
G008.0	Interlock signal for all axes	*IT	○	○
G008.1	Cutting block start interlock signal	*CSL	○	○
G008.3	Block start interlock signal	*BSL	○	○
G008.4	Emergency stop signals	*ESP	○	○
G008.5	Feed hold signal	*SP	○	○
G008.6	Reset & rewind signal	RRW	○	○
G008.7	External reset signal	ERS	○	○
G009.0 to G009.4	External workpiece number search signals	PN1,PN2,PN4,PN8, PN16	○	○
G010,G011	Manual feedrate override signals	*JV0 to *JV15	○	○

Address	Signal name	Symbol	T series	M series
G012	Feedrate override signals	*FV0 to *FV7	○	○
G013	2nd feedrate override signals	*AFV0 to *AFV7	○	○
G014.0,G014.1	Rapid traverse override signals	ROV1,ROV2	○	○
G016.7	One-digit F code feed signal	F1D	-	○
G018.0 to G018.3	Manual handle feed axis selection signals	HS1A to HS1D	○	○
G018.4 to G018.7		HS2A to HS2D	○	○
G019.0 to G019.3		HS3A to HS3D	○	○
G019.4,G019.5, G019.6	Manual handle feed amount selection signals (incremental feed signals)	MP1,MP2,MP4	○	○
G019.7	Manual rapid traverse selection signal	RT	○	○
G020.0 to G020.3	Manual handle feed axis selection signals	HS4A to HS4D	○	○
G021.0 to G022.3	Servo motor rotation speed specification signals	SVR01I to SVR12I	○	○
G022.4	Differential speed synchronization command signal	DFSYC	○	○
G022.5	Servo motor rotation polarity specification signal	SVGN	○	○
G022.7	Servo motor spindle control switching signal	SVSP	○	○
G023.3	Manual handle feed maximum feedrate change signal	HNDLF	○	○
G023.4	Handle-synchronous feed signal	HREV	○	○
G023.5	In-position check disable signal	NOINPS	○	○
G024.0 to G025.5	Extended external workpiece number search signals	EPN0 to EPN13	○	○
G025.7	External workpiece number search start signal	EPNS	○	○
G026.0	Position coder selection signal	PC3SLC	○	○
G026.1		PC4SLC	○	○
G026.3	Spindle selection signals	SWS4	○	○
G026.6	Individual spindle stop signals	*SSTP4	○	○
G027.0	Spindle selection signals	SWS1	○	○
G027.1		SWS2	○	○
G027.2		SWS3	○	○
G027.3	Individual spindle stop signals	*SSTP1	○	○
G027.4		*SSTP2	○	○
G027.5		*SSTP3	○	○
G027.7	Cs contour control change signal	CON	○	○
G028.1,G028.2	Gear selection signals (input)	GR1,GR2	○	○
G028.4	Spindle unclamp completion signal	*SUCPFA	○	○
G028.5	Spindle clamp completion signal	*SCPFA	○	○
G028.6	Spindle stop complete signal	SPSTPA	○	○
G028.7	2nd position coder selection signal	PC2SLC	○	○
G029.0	Gear selection signals (input)	GR21	○	○
G029.1		GR22	○	○
G029.2		GR31	○	○
G029.3		GR32	○	○
G029.4	Spindle speed arrival signal	SAR	○	○
G029.5	Spindle orientation signal	SOR	○	○
G029.6	Spindle stop signal	*SSTP	○	○
G030	Spindle speed override signals	SOV0 to SOV7	○	○
G031.3	Three-dimensional coordinate system conversion manual interruption switch signal	M3R	○	○
G031.4	Gear selection signals (input)	GR41	○	○

Address	Signal name	Symbol	T series	M series
G031.5		GR42	O	O
G031.6	1st spindle parking signal	PKESS1	O	O
G031.7	2nd spindle parking signal	PKESS2	O	O
G032.0 to G033.3	Spindle motor speed command signals	R01I to R12I	O	O
G033.5	Spindle motor command polarity command signals	SGN	O	O
G033.6	Spindle motor command polarity selection signals	SSIN	O	O
G033.7	Spindle motor speed command selection signals	SIND	O	O
G034.0 to G035.3	Spindle motor speed command signals	R01I2 to R12I2	O	O
G035.5	Spindle motor command polarity command signals	SGN2	O	O
G035.6	Spindle motor command polarity selection signals	SSIN2	O	O
G035.7	Spindle motor speed command selection signals	SIND2	O	O
G036.0 to G037.3	Spindle motor speed command signals	R01I3 to R12I3	O	O
G037.5	Spindle motor command polarity command signals	SGN3	O	O
G037.6	Spindle motor command polarity selection signals	SSIN3	O	O
G037.7	Spindle motor speed command selection signals	SIND3	O	O
G038.0	Polygon spindle stop signal	*PLSST	O	O
G038.1	Spindle synchronous speed ratio control signal	SBRT	O	O
G038.2	Spindle synchronous control signal	SPSYC	O	O
G038.3	Spindle phase synchronous control signal	SPPHS	O	O
G038.5	Speed display change signal	SDPC	O	O
G038.6	B axis unclamp completion signal	*BEUCP	-	O
G038.7	B axis clamp completion signal	*BECLP	-	O
G039.0 to G039.5	Tool offset number selection signals	OFN0 to OFN5	O	O
G040.0 to G040.3		OFN6 to OFN9	O	O
G039.6	Workpiece origin offset measurement mode selection signal	WOQSM	-	O
G039.6	Workpiece coordinate system shift value write mode select signal	WOQSM	O	-
G039.7	Tool offset measurement mode selection signal	GOQSM	-	O
G039.7	Tool offset write mode select signal	GOQSM	O	-
G040.5	Spindle measurement select signal	S2TLS	O	-
G040.6	Position record signal	PRC	O	-
G040.7	Workpiece coordinate system shift value write signal	WOSET	O	-
G041.0 to G041.3	Manual handle interrupt axis selection signals	HS1IA to HS1ID	O	O
G041.4 to G041.7		HS2IA to HS2ID	O	O
G042.0 to G042.3		HS3IA to HS3ID	O	O
G042.7	Direct operation select signal	DMMC	O	O
G043.0 to G043.2	Mode selection signals	MD1,MD2,MD4	O	O
G043.5	DNC operation select signal	DNCI	O	O
G043.7	Manual reference position return selection signal	ZRN	O	O
G044.0	Optional block skip signals	BDT1	O	O
G044.1	All-axis machine lock signal	MLK	O	O
G045	Optional block skip signals	BDT2 to BDT9	O	O
G046.0	Memory protection signal	KEYP	O	O

Address	Signal name	Symbol	T series	M series
G046.1	Single block signal	SBK	○	○
G046.3 to G046.6	Memory protection signals	KEY1 to KEY4	○	○
G046.7	Dry run signal	DRN	○	○
G047.0 to G048.1	Tool group number selection signals	TL01 to TL512	○	○
G048.2	Tool life counting disable signal	LFCIV	○	○
G048.5	Tool skip signal	TLSKP	○	○
G048.6	Individual tool change reset signal	TLRSTI	○	○
G048.7	Tool change reset signal	TLRST	○	○
G049.0 to G050.1	Tool life count override signals	*TLV0 to *TLV9	○	○
G051.0 to G051.3	Oscillation feedrate override signals	*CHP1 to *CHP8	○	○
G051.6	Oscillation start signal	CHPST	○	○
G051.7	Oscillation hold signal	*CHLD	○	○
G053.0	General-purpose integrating meter start signal	TMRON	○	○
G053.3	Interrupt signal for custom macro	UINT	○	○
G053.5	Rapid traverse block overlap disable signal	ROVLP	○	○
G053.6	In-position check signal	SMZ	○	○
G053.7	Chamfering signal	*CDZ	○	-
G054 to G057	Custom macro input signals	UI000 to UI031	○	○
G058.1	External input start signal	EXINP	○	○
G058.2	External input/output stop signal	EXSTP	○	○
G058.3	External output start signal	EXOUT	○	○
G059.0	Tool retraction signal	TRESC	○	○
G059.1	Tool return signal	TRRTN	○	○
G059.7	Signal for disabling torque difference alarm detection for axis synchronous control	NSYNCA	○	○
G060.7	Tail stock barrier selection signal	*TSB	○	-
G061.0	Rigid tapping signal	RGTAP	○	○
G061.2	Servo motor spindle synchronization start signal	SYSS	○	○
G061.4 to G061.7	Rigid tapping spindle selection signals	RGTSP1 to RGTSP4	○	-
G062.1	Screen erasure disable signal	*CRTOF	○	○
G062.6	Rigid tapping retraction start signal	RTNT	○	○
G062.7	Path select signal（Tool post select signal） 2	HEAD2	●	●
G063.0	Path select signal (Tool post select signal)	HEAD	●	●
G063.1	No-wait signal	NOWT	●	●
G063.2,G063.3	Path spindle command selection signals	SLSPA,SLSPB	●	●
G063.5	Signal for disabling angular axis control for the perpendicular axis	NOZAGC	○	○
G063.6	In-feed control cut start signal	INFD	-	○
G063.7	No-wait signal	NMWT	●	●
G064.2,G064.3	Path spindle feedback selection signals	SLPCA,SLPCB	●	●
G064.6	Spindle command synchronous control signal	ESRSYC	○	○
G066.0	All-axis VRDY off alarm ignore signal	IGNVRY	○	○
G066.1	External key input mode selection signal	ENBKY	○	○
G066.4	Retract signal	RTRCT	○	○
G066.7	Key code read signal	EKSET	○	○
G067.0	Manual tool compensation command number	MTLC	○	-
G067.2	Checking mode signal	MMOD	○	○
G067.3	Handle available signal in checking mode	MCHK	○	○
G067.4	EGB synchronization mode selection signal	EGBS	○	○

Address	Signal name	Symbol	T series	M series
G067.6	Hard copy cancellation request signal	HCABT	○	○
G067.7	Hard copy execution request signal	HCREQ	○	○
G68,G69	Manual tool compensation tool number signal (4 digits)	MTLN00 to MTLN15	○	-
G070.0	Torque limit command LOW signals (serial spindle)	TLMLA	○	○
G070.1	Torque limit command HIGH signals (serial spindle)	TLMHA	○	○
G070.3,G070.2	Clutch/gear signals(serial spindle)	CTH1A,CTH2A	○	○
G070.4	CCW command signals(serial spindle)	SRVA	○	○
G070.5	CW command signals(serial spindle)	SFRA	○	○
G070.6	Orientation command signals (serial spindle)	ORCMA	○	○
G070.7	Machine ready signals(serial spindle)	MRDYA	○	○
G071.0	Alarm reset signals (serial spindle)	ARSTA	○	○
G071.1	Emergency stop signals(serial spindle)	*ESPA	○	○
G071.2	Spindle selection signals (serial spindle)	SPSLA	○	○
G071.3	Power line switch completion signals (serial spindle)	MCFNA	○	○
G071.4	Soft start/stop signals(serial spindle)	SOCNA	○	○
G071.5	Speed integral signals (serial spindle)	INTGA	○	○
G071.6	Output switch request signals (serial spindle)	RSLA	○	○
G071.7	Power line status check signals (serial spindle)	RCHA	○	○
G072.0	Orientation stop position change command signals (serial spindle)	INDXA	○	○
G072.1	Rotational direction command signals for orientation stop position change (serial spindle)	ROTAA	○	○
G072.2	Shortcut command signals for orientation stop position change (serial spindle)	NRROA	○	○
G072.3	Differential speed mode command signals (serial spindle)	DEFMDA	○	○
G072.4	Analog override signals (serial spindle)	OVRA	○	○
G072.5	Incremental command externally set orientation signals(serial spindle)	INCMDA	○	○
G072.6	Spindle switch MAIN MCC contact status signals(serial spindle)	MFNHGA	○	○
G072.7	Spindle switch HIGH MCC contact status signals (serial spindle)	RCHHGA	○	○
G073.0	Magnetic sensor orientation command signal(serial spindle)	MORCMA	○	○
G073.1	Subordinate operation mode command signals (serial spindle)	SLVA	○	○
G073.2	Motor power cutoff command signals (serial spindle)	MPOFA	○	○
G073.3	Synchronous orientation request command	SORSLA	○	○
G073.4	Disconnection detection disable signal (serial spindle)	DSCNA	○	○
G074.0	Torque limit command LOW signals (serial spindle)	TLMLB	○	○
G074.1	Torque limit command HIGH signals (serial spindle)	TLMHB	○	○
G074.3,G074.2	Clutch/gear signals(serial spindle)	CTH1B,CTH2B	○	○
G074.4	CCW command signals(serial spindle)	SRVB	○	○
G074.5	CW command signals(serial spindle)	SFRB	○	○
G074.6	Orientation command signals (serial spindle)	ORCMB	○	○
G074.7	Machine ready signals(serial spindle)	MRDYB	○	○
G075.0	Alarm reset signals (serial spindle)	ARSTB	○	○
G075.1	Emergency stop signals(serial spindle)	*ESPB	○	○
G075.2	Spindle selection signals (serial spindle)	SPSLB	○	○
G075.3	Power line switch completion signals (serial spindle)	MCFNB	○	○
G075.4	Soft start/stop signals(serial spindle)	SOCNB	○	○

Address	Signal name	Symbol	T series	M series
G075.5	Speed integral signals (serial spindle)	INTGB	○	○
G075.6	Output switch request signals (serial spindle)	RSLB	○	○
G075.7	Power line status check signals (serial spindle)	RCHB	○	○
G076.0	Orientation stop position change command signals (serial spindle)	INDXB	○	○
G076.1	Rotational direction command signals for orientation stop position change (serial spindle)	ROTAB	○	○
G076.2	Shortcut command signals for orientation stop position change (serial spindle)	NRROB	○	○
G076.3	Differential speed mode command signals (serial spindle)	DEFMDB	○	○
G076.4	Analog override signals (serial spindle)	OVRB	○	○
G076.5	Incremental command externally set orientation signals(serial spindle)	INCMDB	○	○
G076.6	Spindle switch MAIN MCC contact status signals (serial spindle)	MFNHGB	○	○
G076.7	Spindle switch HIGH MCC contact status signals (serial spindle)	RCHHGB	○	○
G077.0	Magnetic sensor orientation command signal(serial spindle)	MORCMB	○	○
G077.1	Subordinate operation mode command signals (serial spindle)	SLVB	○	○
G077.2	Motor power cutoff command signals (serial spindle)	MPOFB	○	○
G077.3	Synchronous orientation request command	SORSLB	○	○
G077.4	Disconnection detection disable signal (serial spindle)	DSCNB	○	○
G078.0 to G079.6	Spindle orientation signals with the stop position externally set	SH00A to SH14A	○	○
G080.0 to G081.6		SH00B to SH14B	○	○
G082,G083	Input signals for P-code macro	EUI00 to EUI15	○	○
G086.0 to G086.3	Feed axis and direction selection signals	+Jg, -Jg, +Ja, -Ja	○	○
G087.0,G087.1	Manual handle feed amount selection signals(Incremental feed signals)	MP21,MP22	○	○
G087.3,G087.4		MP31,MP32	○	○
G087.6,G087.7		MP41,MP42	○	○
G088.3	Manual pulse magnification change signal	HNDMP	○	○
G088.4 to G088.7	Manual handle interrupt axis selection signals	HS4IA to HS4ID	○	○
G090.0	Tool offset direction signals	G2RVX	○	-
G090.1		G2RVZ	○	-
G090.2		G2RVY	○	-
G090.4	2nd geometry tool offset axis select signals	G2X	○	-
G090.5		G2Z	○	-
G090.6		G2Y	○	-
G090.7	2nd geometry tool offset signal	G2SLC	○	-
G096.0 to G096.6	1% rapid traverse override signals	*HROV0 to *HROV6	○	○
G096.7	1% step rapid traverse override selection signal	HROV	○	○
G098	Key code signals	EKC0 to EKC7	○	○
G100	Feed axis and direction selection signals	+J1 to +J8	○	○
G101	External deceleration signals 2	*+ED21 to *+ED28	○	○
G102	Feed axis and direction selection signals	-J1 to -J8	○	○
G103	External deceleration signals 2	*-ED21 to *-ED28	○	○

Address	Signal name	Symbol	T series	M series
G104	Axis direction dependent stored stroke check 1 switch signals	+EXL1 to +EXL8	O	O
G105		-EXL1 to -EXL8	O	O
G106	Mirror image signals	MI1 to MI8	O	O
G107	External deceleration signals 3	*+ED31 to *+ED38	O	O
G108	Each-axis machine lock signals	MLK1 to MLK8	O	O
G109	External deceleration signals 3	*-ED31 to *-ED38	O	O
G110	Stroke limit external setting signals	+LM1 to +LM8	O	O
G112		-LM1 to -LM8	O	O
G114	Overtravel signals	*+L1 to *+L8	O	O
G116		*-L1 to *-L8	O	O
G118	External deceleration signals 1	*+ED1 to *+ED8	O	O
G120		*-ED1 to *-ED8	O	O
G122	Parking signals	PK1 to PK8	O	O
G122.6(G031.6)	1st spindle parking signal	PKESS1	O	O
G122.7(G031.7)	2nd spindle parking signal	PKESS2	O	O
G124	Controlled axis detach signals	DTCH1 to DTCH8	O	O
G125	Unexpected disturbance torque detection ignore signal	IUDD1 to IUDD8	O	O
G126	Servo off signals	SVF1 to SVF8	O	O
G128	Composite control axis change selection signals	MIX1 to MIX8	O	O
G130	Interlock signal for each axis	*IT1 to *IT8	O	O
G132	Interlock signal for each axis direction	+MIT1 to +MIT8	-	O
G134		-MIT1 to -MIT8		
G132.0,G132.1	Tool offset write signals	+MIT1,+MIT2	O	-
G134.0,G134.1	Tool offset write signals	-MIT1,-MIT2	O	-
G132.0	Tool offset write signals	+MIT1	-	O
G136	Control axis selection signals (PMC axis control)	EAX1 to EAX8	O	O
G138	Synchronous control axis selection signals	SYNC1 to SYNC8	O	O
G140	Signals for selecting the manual feed axis for axis synchronous control	SYNCJ1 to SYNCJ8	O	O
G142.0	Auxiliary function completion signal (for group 1) (PMC axis control)	EFINA	O	O
G142.1	Accumulated zero check signal (for group 1) (PMC axis control)	ELCKZA	O	O
G142.2	Buffering disable signal (for group 1) (PMC axis control)	EMBUFA	O	O
G142.3	Block stop signal (for group 1) (PMC axis control)	ESBKA	O	O
G142.4	Servo-off signal (for group 1) (PMC axis control)	ESOFA	O	O
G142.5	Axis control temporary stop signal (for group 1) (PMC axis control)	ESTPA	O	O
G142.6	Reset signal (for group 1) (PMC axis control)	ECLRA	O	O
G142.7	Axis control command read signal (for group 1) (PMC axis control)	EBUFA	O	O
G143.0 to G143.6	Axis control command signals (for group 1) (PMC axis control)	EC0A to EC6A	O	O
G143.7	Block stop disable signal (for group 1) (PMC axis control)	EMSBKA	O	O
G144,G145	Axis control feedrate signals (for group 1) (PMC axis control)	EIF0A to EIF15A	O	O
G146 to G149	Axis control data signals (for group 1) (PMC axis control)	EID0A to EID31A	O	O

Address	Signal name	Symbol	T series	M series
G150.0,G150.1	Rapid traverse override signals (PMC axis control)	EROV1,EROV2	○	○
G150.5	Override cancellation signal (for group 1) (PMC axis control)	EOVC	○	○
G150.6	Manual rapid traverse selection signal (PMC axis control)	ERT	○	○
G150.7	Dry run signal (PMC axis control)	EDRN	○	○
G151	Feedrate override signals (for group 1) (PMC axis control)	*EFOV0 to *EFOV7	○	○
G151	1% step rapid traverse override signals (for group 1) (PMC axis control)	*EROV0 to *EROV7	○	○
G154.0	Auxiliary function completion signal (for group 2) (PMC axis control)	EFINB	○	○
G154.1	Accumulated zero check signal (for group 2) (PMC axis control)	ELCKZB	○	○
G154.2	Buffering disable signal (for group 2) (PMC axis control)	EMBUFB	○	○
G154.3	Block stop signal (for group 2) (PMC axis control)	ESBKB	○	○
G154.4	Servo-off signal (for group 2) (PMC axis control)	ESOFB	○	○
G154.5	Axis control temporary stop signal (for group 2) (PMC axis control)	ESTPB	○	○
G154.6	Reset signal (for group 2) (PMC axis control)	ECLRB	○	○
G154.7	Axis control command read signal (for group 2) (PMC axis control)	EBUFB	○	○
G155.0 to G155.6	Axis control command signals (for group 2) (PMC axis control)	EC0B to EC6B	○	○
G155.7	Block stop disable signal (for group 2) (PMC axis control)	EMSBKB	○	○
G156,G157	Axis control feedrate signals (for group 2) (PMC axis control)	EIF0B to EIF15B	○	○
G158 to G161	Axis control data signals (for group 2) (PMC axis control)	EID0B to EID31B	○	○
G162.5	Override cancellation signal (for group 2) (PMC axis control)	EOVCB	○	○
G163	Feedrate override signals (for group 2) (PMC axis control)	*EFOV0B to *EFOV7B	○	○
G163	1% step rapid traverse override signals (for group 2) (PMC axis control)	*EROV0B to *EROV7B	○	○
G166.0	Auxiliary function completion signal (for group 3) (PMC axis control)	EFINC	○	○
G166.1	Accumulated zero check signal (for group 3) (PMC axis control)	ELCKZC	○	○
G166.2	Buffering disable signal (for group 3) (PMC axis control)	EMBUFC	○	○
G166.3	Block stop signal (for group 3) (PMC axis control)	ESBKC	○	○
G166.4	Servo-off signal (for group 3) (PMC axis control)	ESOFC	○	○
G166.5	Axis control temporary stop signal (for group 3) (PMC axis control)	ESTPC	○	○
G166.6	Reset signal (for group 3) (PMC axis control)	ECLRC	○	○

Address	Signal name	Symbol	T series	M series
G166.7	Axis control command read signal (for group 3) (PMC axis control)	EBUFC	○	○
G167.0 to G167.6	Axis control command signals (for group 3) (PMC axis control)	EC0C to EC6C	○	○
G167.7	Block stop disable signal (for group 3) (PMC axis control)	EMSBKC	○	○
G168,G169	Axis control feedrate signals (for group 3) (PMC axis control)	EIF0C to EIF15C	○	○
G170 to G173	Axis control data signals (for group 3) (PMC axis control)	EID0C to EID31C	○	○
G174.5	Override cancellation signal (for group 3) (PMC axis control)	EOVCC	○	○
G175	Feedrate override signals (for group 3) (PMC axis control)	*EFOV0C to *EFOV7C	○	○
G175	1% step rapid traverse override signals (for group 3) (PMC axis control)	*EROV0C to *EROV7C	○	○
G178.0	Auxiliary function completion signal (for group 4) (PMC axis control)	EFIND	○	○
G178.1	Accumulated zero check signal (for group 4) (PMC axis control)	ELCKZD	○	○
G178.2	Buffering disable signal (for group 4) (PMC axis control)	EMBUFD	○	○
G178.3	Block stop signal (for group 4) (PMC axis control)	ESBKD	○	○
G178.4	Servo-off signal (for group 4) (PMC axis control)	ESOFD	○	○
G178.5	Axis control temporary stop signal (for group 4) (PMC axis control)	ESTPD	○	○
G178.6	Reset signal (for group 4) (PMC axis control)	ECLRD	○	○
G178.7	Axis control command read signal (for group 4) (PMC axis control)	EBUFD	○	○
G179.0 to G179.6	Axis control command signals (for group 4) (PMC axis control)	EC0D to EC6D	○	○
G179.7	Block stop disable signal (for group 4) (PMC axis control)	EMSBKD	○	○
G180,G181	Axis control feedrate signals (for group 4) (PMC axis control)	EIF0D to EIF15D	○	○
G182 to G185	Axis control data signals (for group 4) (PMC axis control)	EID0D to EID31D	○	○
G186.5	Override cancellation signal (for group 4) (PMC axis control)	EOVCD	○	○
G187	Feedrate override signals (for group 4) (PMC axis control)	*EFOV0D to *EFOV7D	○	○
G187	1% step rapid traverse override signals (for group 4) (PMC axis control)	*EROV0D to *EROV7D	○	○
G190	Superimposed control axis selection signals	OVLS1 to OVLS8	○	○
G192	Each-axis VRDY off alarm ignore signal	IGVRY1 to IGVRY8	○	○
G193.3	Selecting direction of manual handle rotation signal	HDSR	○	○
G196	Reference position return deceleration signals	*DEC1 to *DEC8	○	○
G197.0 to G197.3	Flexible synchronization control mode selection signals	MTA,MTB,MTC,MTD	○	○
G198	Axis non-displayed signals	NPOS1 to NPOS8	○	○
G199.0,G199.1	Manual handle generators selection signal	IOLBH1, IOLBH2	○	○

Address	Signal name	Symbol	T series	M series
G202	A/B phase detector disconnection alarm ignore signal (PMC axis control)	NDCAL1 to NDCAL8	○	○
G203.3	Axis immediate stop start signal	ESTPR	○	○
G203.7	Power failure deceleration signal	PWFL	○	○
G204.0	Torque limit command LOW signals (serial spindle)	TLMLC	○	○
G204.1	Torque limit command HIGH signals (serial spindle)	TLMHC	○	○
G204.3,G204.2	Clutch/gear signals(serial spindle)	CTH1C,CTH2C	○	○
G204.4	CCW command signals(serial spindle)	SRVC	○	○
G204.5	CW command signals(serial spindle)	SFRC	○	○
G204.6	Orientation command signals (serial spindle)	ORCMC	○	○
G204.7	Machine ready signals(serial spindle)	MRDYC	○	○
G205.0	Alarm reset signals(serial spindle)	ARSTC	○	○
G205.1	Emergency stop signals(serial spindle)	*ESPC	○	○
G205.2	Spindle selection signals (serial spindle)	SPSLC	○	○
G205.3	Power line switch completion signals (serial spindle)	MCFNC	○	○
G205.4	Soft start/stop signals (serial spindle)	SOCNC	○	○
G205.5	Speed integral signals (serial spindle)	INTGC	○	○
G205.6	Output switch request signals (serial spindle)	RSLC	○	○
G205.7	Power line status check signals (serial spindle)	RCHC	○	○
G206.0	Orientation stop position change command signals (serial spindle)	INDXC	○	○
G206.1	Rotational direction command signals for orientation stop position change (serial spindle)	ROTAC	○	○
G206.2	Shortcut command signals for orientation stop position change (serial spindle)	NRROC	○	○
G206.3	Differential speed mode command signals (serial spindle)	DEFMDC	○	○
G206.4	Analog override signals (serial spindle)	OVRC	○	○
G206.5	Incremental command externally set orientation signals(serial spindle)	INCMDC	○	○
G206.6	Spindle switch MAIN MCC contact status signals (serial spindle)	MFNHGC	○	○
G206.7	Spindle switch HIGH MCC contact status signals (serial spindle)	RCHHGC	○	○
G207.0	Magnetic sensor orientation command signal(serial spindle)	MORCMC	○	○
G207.1	Subordinate operation mode command signals (serial spindle)	SLVC	○	○
G207.2	Motor power cutoff command signals (serial spindle)	MPOFC	○	○
G207.3	Synchronous orientation request command	SORSLC	○	○
G207.4	Disconnection detection disable signal (serial spindle)	DSCNC	○	○
G208.0 to G209.6	Spindle orientation signals with the stop position externally set	SH00C to SH14C	○	○
G210 to G211	Data signals for external data input	ED31 to ED16	○	○
G264.0 to G264.3	Spindle command synchronous control signal (for each spindle)	ESSYC1 to ESSYC4	○	○
G265.0 to G265.3	Spindle command synchronous parking signal (for each spindle)	PKESE1 to PKESE4	○	○
G266.0	Torque limit command LOW signals (serial spindle)	TLMLD	○	○
G266.1	Torque limit command HIGH signals (serial spindle)	TLMHD	○	○
G266.2,G266.3	Clutch/gear signals(serial spindle)	CTH2D, CTH1D	○	○
G266.4	CCW command signals(serial spindle)	SRVD	○	○
G266.5	CW command signals(serial spindle)	SFRD	○	○

Address	Signal name	Symbol	T series	M series
G266.6	Orientation command signals (serial spindle)	ORCMD	○	○
G266.7	Machine ready signals(serial spindle)	MRDYD	○	○
G267.0	Alarm reset signals (serial spindle)	ARSTD	○	○
G267.1	Emergency stop signals(serial spindle)	*ESPD	○	○
G267.2	Spindle selection signals (serial spindle)	SPSLD	○	○
G267.3	Power line switch completion signals (serial spindle)	MCFND	○	○
G267.4	Soft start/stop signals (serial spindle)	SOCND	○	○
G267.5	Speed integral signals(serial spindle)	INTGD	○	○
G267.6	Output switch request signals (serial spindle)	RSLD	○	○
G267.7	Power line status check signals (serial spindle)	RCHD	○	○
G268.0	Orientation stop position change command signals(serial spindle)	INDXD	○	○
G268.1	Rotational direction command signals for orientation stop position change (serial spindle)	ROTAD	○	○
G268.2	Shortcut command signals for orientation stop position change (serial spindle)	NRROD	○	○
G268.3	Differential speed mode command signals (serial spindle)	DEFMDD	○	○
G268.4	Analog override signals (serial spindle)	OVRD	○	○
G268.5	Incremental command externally set orientation signals(serial spindle)	INCMDD	○	○
G268.6	Spindle switch MAIN MCC contact status signals (serial spindle)	MFNHGD	○	○
G268.7	Spindle switch HIGH MCC contact status signals (serial spindle)	RCHHGD	○	○
G269.0	Magnetic sensor orientation command signal(serial spindle)	MORCMD	○	○
G269.1	Subordinate operation mode command signals (serial spindle)	SLVD	○	○
G269.2	Motor power cutoff command signals (serial spindle)	MPOFD	○	○
G269.3	Synchronous orientation request command	SORSLD	○	○
G269.4	Disconnection detection disable signal (serial spindle)	DSCND	○	○
G270.0 to G271.6	Spindle orientation signals with the stop position externally set	SH00D to SH14D	○	○
G272.0 to G273.3	Spindle motor speed command signals	R01I4 to R12I4	○	○
G273.5	Spindle motor command polarity command signals	SGN4	○	○
G273.6	Spindle motor command polarity selection signals	SSIN4	○	○
G273.7	Spindle motor speed command selection signals	SIND4	○	○
G274.0 to G274.3	Cs contour control change signal (for each spindle)	CONS1 to CONS4	○	○
G274.4 to G274.7	Cs axis coordinate establishment request signals (for each spindle)	CSFI1 to CSFI4	○	○
G276 to G279	Custom macro input signals	UI100 to UI131	○	○
G280 to G283		UI200 to UI231	○	○
G284 to G287		UI300 to UI331	○	○
G288.0 to G288.3	Spindle synchronous control signal (for each spindle)	SPSYC1 to SPSYC4	○	○
G289.0 to G289.3	Spindle phase synchronous control signal (for each spindle)	SPPHS1 to SPPHS4	○	○
G290.5	High-speed program check signal	PGCK	○	○
G295.6	Dual display forcible end request signal	C2SEND	○	○
G295.7	Key control selection signal	CNCKY	○	○

Address	Signal name	Symbol	T series	M series
G296	Diameter/radius specification switch signals (each axis)	DI1 to DI8	○	○
G297.0	Block cancel signal	BCAN	○	○
G299.7	Reset key input invalid signal	IRTKY	○	○
G304.3	Reference position establishment starting signal (serial spindle)	CSYCA	○	○
G304.6	Inertia estimation start signal (serial spindle)	INESTRA	○	○
G304.7	Adaptive resonance elimination filter search mode signal (serial spindle)	FRFSMA	○	○
G305.0 to G305.3	Resonance elimination filter disable signal (serial spindle)	HF1A to HF4A	○	○
G306.1	Cutting feed/rapid traverse PWM frequency switching function in Cs contour control enable signal (serial spindle)	PWMSEA	○	○
G306.2	Preload and multi-axis integrator copy disable signal (serial spindle)	TDFCANA	○	○
G308.3	Reference position establishment starting signal (serial spindle)	CSYCB	○	○
G308.6	Inertia estimation start signal (serial spindle)	INESTRB	○	○
G308.7	Adaptive resonance elimination filter search mode signal (serial spindle)	FRFSMB	○	○
G309.0 to G309.3	Resonance elimination filter disable signal (serial spindle)	HF1B to HF4B	○	○
G310.1	Cutting feed/rapid traverse PWM frequency switching function in Cs contour control enable signal (serial spindle)	PWMSEB	○	○
G310.2	Preload and multi-axis integrator copy disable signal (serial spindle)	TDFCANB	○	○
G312.3	Reference position establishment starting signal (serial spindle)	CSYCC	○	○
G312.6	Inertia estimation start signal (serial spindle)	INESTRC	○	○
G312.7	Adaptive resonance elimination filter search mode signal (serial spindle)	FRFSMC	○	○
G313.0 to G313.3	Resonance elimination filter disable signal (serial spindle)	HF1C to HF4C	○	○
G314.1	Cutting feed/rapid traverse PWM frequency switching function in Cs contour control enable signal (serial spindle)	PWMSEC	○	○
G314.2	Preload and multi-axis integrator copy disable signal (serial spindle)	TDFCANC	○	○
G316.3	Reference position establishment starting signal (serial spindle)	CSYCD	○	○
G316.6	Inertia estimation start signal (serial spindle)	INESTRD	○	○
G316.7	Adaptive resonance elimination filter search mode signal (serial spindle)	FRFSMD	○	○
G317.0 to G317.3	Resonance elimination filter disable signal (serial spindle)	HF1D to HF4D	○	○
G318.1	Cutting feed/rapid traverse PWM frequency switching function in Cs contour control enable signal (serial spindle)	PWMSED	○	○
G318.2	Preload and multi-axis integrator copy disable signal (serial spindle)	TDFCAND	○	○
G328.0 to G328.3	Tool change reset signals 1 to 4	TLRST1 to TLRST4	○	○

Address	Signal name	Symbol	T series	M series
G328.4 to G328.7	Individual tool change reset signals 1 to 4	TLRSTI1 to TLRSTI4	○	○
G329.0 to G329.3	Tool skip signals 1 to 4	TLSKP1 to TLSKP4	○	○
G329.4 to G329.7	Tool life counting disable signals 1 to 4	TLNCT1 to TLNCT4	○	○
G330.0 to G330.5	Tool management data protection signal	TKEY0 to TKEY5	○	○
G340.5	Manual 2nd/3rd/4th reference position return select 1 signal	SLREF	○	○
G340.6	Manual 2nd/3rd/4th reference position return select 2 signal	SLRER	○	○
G341	External deceleration signals 4	*+ED41 to *+ED48	○	○
G342		*-ED41 to *-ED48	○	○
G343	External deceleration signals 5	*+ED51 to *+ED58	○	○
G344		*-ED51 to *-ED58	○	○
G347.1	Manual handle feed direction inversion signal	HDN	○	○
G347.7	3-dimensional coordinate system conversion manual interrupt enable/disable switch signal	NOT3DM	○	○
G351.0 to G351.3	Simple spindle EGB signals	SSEGB1 to SSEGB4	○	○
G352.0 to G353.1	0.1% rapid traverse override signals	*FHRO0 to *FHRO9	○	○
G353.7	0.1% step rapid traverse override selection signal	FHROV	○	○
G358	Each axis workpiece coordinate system preset signals	WPRST1 to WPRST8	○	○
G376	2nd spindle speed override signals	SOV20 to SOV27	○	○
G377	3rd spindle speed override signals	SOV30 to SOV37	○	○
G378	4th spindle speed override signals	SOV40 to SOV47	○	○
G379.0 to G379.3	Manual handle feed axis selection signals	HS5A to HS5D	○	○
G379.4 to G379.7	Manual handle interrupt axis selection signals	HS5IA to HS5ID	○	○
G380.0,G380.1	Manual handle feed amount selection signals	MP51,MP52	○	○
G381.0 to G381.3	Flexible synchronization control automatic phase synchronization signals	AUTPHA to AUTPHD	○	○
G400.1	Spindle unclamp completion signal	*SUCPFB	○	○
G400.2		*SUCPFC	○	○
G400.3		*SUCPFD	○	○
G401.1	Spindle clamp completion signal	*SCPFB	○	○
G401.2		*SCPFC	○	○
G401.3		*SCPFD	○	○
G402.1	Spindle stop complete signal	SPSTPB	○	○
G402.2		SPSTPC	○	○
G402.3		SPSTPD	○	○
G403.0,G403.1	Path spindle command selection signals	SLSPC,SLSPD	●	●
G403.4,G403.5	Path spindle feedback selection signals	SLPCC,SLPCD	●	●
G406.0 to G407.1	Path interference check association signal	ITF01 to ITF10	●	-
G408.0	Start check signal	STCHK	○	○
G408.1,G408.2	Path select signals 3,4	HEAD3,4	●	●
G411.0 to G411.3	Manual handle feed axis selection signals	HS1E to HS4E	○	○
G411.4 to G411.7	Manual handle interrupt axis selection signals	HS1IE to HS4IE	○	○
G412.0	Manual handle feed axis selection signals	HS5E	○	○

Address	Signal name	Symbol	T series	M series
G412.4	Manual handle interrupt axis selection signals	HS5IE	○	○
G512,G513	Macro call start signal	MCST1 to MCST16	○	○
G514.0	Mode change completion signal	MCFIN	○	○
G514.4	All programs save request signal	HPMRSV	○	○
G517.0	Measuring position reached signals	GAE1	○	○
G517.1		GAE2	○	○
G517.2		GAE3	-	○
G517.6	Program restart memory storing disabled signal	QRSTD	○	○
G517.7	Phase synchronization for Servo/Spindle synchronous start signal	SYPST	○	○
G518.4	DeviceNet communication error clear signal	DNTCLR	○	○
G521	SV speed control mode signals	SRVON1 to SRVON8	○	○
G523	SV reverse signals	SVRVS1 to SVRVS8	○	○
G525 to G528	Manual tool compensation tool number signal (8 digits)	MT8N00 to MT8N31	○	-
G530	EGB synchronization start signals	EGBS1 to EGBS8	○	○
G531.0	Forward movement prohibition signal	FWSTP	○	○
G531.1	Reverse movement prohibition signal	MRVM	○	○
G531.3	Dual position feedback turning mode selection signal	HBTRN	○	○
G531.4	Inter-path flexible synchronization mode select signal Advanced superimposition signal AI contour control permission signal	OVLN	○	○
G531.6, G531.7	Stored stroke check 1 select signals	EXLM2, EXLM3	○	○
G533.0 to G533.3	Total spindle revolution number reset signals	SSR1 to SSR4	○	○
G533.4	Total spindle revolution number reset selection signal	SSRS	○	○
G534.0	Axis switching signals	AXC1	○	○
G534.1		AXC2	○	○
G534.2		AXC4	○	○
G536.2	Removal start signal	RMVST	○	○
G536.3	Assignment start signal	ASNST	○	○
G536.4	Exchange start signal	EXCST	○	○
G536.5	Direct assignment mode signal	DASN	○	○
G536.7	Spindle command path specification signal	SPSP	○	○
G544.0 to G544.4	Manual linear/circular interpolation signals	MHLC1 to MHLC5	○	○
G545.0 to G545.4	Usage selection of manual linear/circular interpolation signals	MHUS1 to MHUS5	○	○
G546.7	Tool offset write mode select signal (for milling and turning function)	GQSMC	-	○
G546.0 to G546.5, G547.0 to G547.3	Tool offset number selection signals (for milling and turning function)	OFNC0 to OFNC5, OFNC6 to OFNC9	-	○
G547.6	Tool compensation number specification signal	ONSC	○	–
G548	Dual position feedback compensation clamp signals	*CL1 to *CL8	○	○
G549.0 to G549.3	Cs contour control high speed switching signals	CONH1 to CONH4	○	○
G549.4	Groove of thread measurement signal	GTMSR	○	○
G549.5	Re-machining thread signal	RMTC	○	○
G549.6	Chamfering for arbitrary speed threading signal	ASTC	○	○

Address	Signal name	Symbol	T series	M series
G579.5	Web browser connection prohibition signal	WBEND	○	○
G579.6	Waiting M codes of high-speed type invalid signal	NHSW	●	●
	Speed-up of non-buffering command by G code invalid signal		○	○
G580	Actual speed display axis selection signals	*ACTF1 to *ACTF8	○	○
G581.0 to G581.6	Display language setting signals	LANG1 to LANG7	○	○
G581.7	Display language switch start signal	SLANG	○	○
G586.4 to G586.7	Spindle control mode off signals	MDOFF1 to MDOFF4	○	○
G587.0 to G587.3	Spindle position save start signal	SPMST1 to SPMST4	○	○
G587.4 to G587.7	Arbitrary spindle position phase synchronization signal	SPAPH1 to SPAPH4	○	○
G588.0 to G588.3	Spindle position save selection signal	SMSL11 to SMSL14	○	○
G588.4 to G588.7		SMSL21 to SMSL24	○	○
G594 to G595	Stored stroke limit range switching data selection signals	OTD0 to OTD15	○	○
G596	Stored stroke limit range switching axis selection signals	OTA1 to OTA8	○	○
G597	Stored stroke limit range switching selection signals	+OT11, -OT11, +OT12, -OT12, +OT2, -OT2, +OT3, -OT3	○	○
G598	Stored stroke limit range switching cancellation signals	+OT11C, -OT11C, +OT12C, -OT12C, +OT2C, -OT2C, +OT3C, -OT3C	○	○
G599.0	Stored stroke limit range switching start signal	OTSW	○	○
G599.3	Servo loop gain / in-position width switching signal	GIS	○	○
G599.4 to G599.5	Time constant of acceleration / deceleration after interpolation for cutting feed switching signals	CTC2 to CTC3	○	○
G599.6 to G599.7	Time constant of acceleration / deceleration after interpolation for rapid traverse switching signals	RTC2 to RTC3	○	○
G687	Dual control axes switching signal	SVMWC1 to SVMWC8	○	○
G708 to G711	Extended spindle motor speed command signals	RE01I to RE32I	○	○
G712 to G715		RE01I2 to RE32I2	○	○
G716 to G719		RE01I3 to RE32I3	○	○
G720 to G723		RE01I4 to RE32I4	○	○
G726	Total travel distance clear signal	TDC1 to TDC8	○	○
G765	DI signal for Data transfer between PMC and DCSPMC	TPMG00 to TPMG07	○	○
F000.0	Rewinding signal	RWD	○	○
F000.4	Feed hold lamp signal	SPL	○	○
F000.5	Cycle start lamp signal	STL	○	○
F000.6	Servo ready signal	SA	○	○
F000.7	Automatic operation signal	OP	○	○
F001.0	Alarm signal	AL	○	○
F001.1	Resetting signal	RST	○	○
F001.2	Battery alarm signal	BAL	○	○
F001.3	Distribution completion signals	DEN	○	○

Address	Signal name	Symbol	T series	M series
F001.4	Spindle enable signal	ENB	○	○
F001.5	Tapping signal	TAP	○	○
F001.7	CNC ready signal	MA	○	○
F002.0	Inch input signal	INCH	○	○
F002.1	Rapid traversing signal	RPDO	○	○
F002.2	Constant surface speed signal	CSS	○	○
F002.3	Threading signal	THRD	○	○
F002.4	Program restart under way signal	SRNMV	○	○
F002.6	Cutting feed signal	CUT	○	○
F002.7	Dry run check signal	MDRN	○	○
F003.0	Incremental feed selection check signal	MINC	○	○
F003.1	Manual handle feed selection check signal	MH	○	○
F003.2	Jog feed selection check signal	MJ	○	○
F003.3	Manual data input selection check signal	MMDI	○	○
F003.4	DNC operation selection confirm signal	MRMT	○	○
F003.5	Automatic operation selection check signal	MMEM	○	○
F003.6	Program edit selection check signal	MEDT	○	○
F004.0	Optional block skip check signals	MBDT1	○	○
F004.1	All-axis machine lock check signal	MMLK	○	○
F004.2	Manual absolute check signal	MABSM	○	○
F004.3	Single block check signal	MSBK	○	○
F004.4	Auxiliary function lock check signal	MAFL	○	○
F004.5	Manual reference position return selection check signal	MREF	○	○
F005	Optional block skip check signals	MBDT2 to MBDT9	○	○
F006.0	Touch panel check signal	TPPRS	○	○
F006.1	MDI reset confirmation signal	MDIRST	○	○
F006.2	Automatic screen erasure status in-progress signal	ERTVA	○	○
F007.0	Auxiliary function strobe signals	MF	○	○
F007.2	Spindle function strobe signal	SF	○	○
F007.3	Tool function strobe signal	TF	○	○
F007.7	2nd auxiliary function strobe signal	BF	○	○
F008.4	2nd M function strobe signal	MF2	○	○
F008.5	3rd M function strobe signal	MF3	○	○
F008.6	4th M function strobe signal	MF4	○	○
F008.7	5th M function strobe signal	MF5	○	○
F009.4	Decode M signals	DM30	○	○
F009.5		DM02	○	○
F009.6		DM01	○	○
F009.7		DM00	○	○
F010 to F013	Auxiliary function code signals	M00 to M31	○	○
F014 to F015	2nd M function code signals	M200 to M215	○	○
F016 to F017	3rd M function code signals	M300 to M315	○	○
	2nd M function code signals	M216 to M231	○	○
F022 to F025	Spindle function code signals	S00 to S31	○	○
F026 to F029	Tool function code signals	T00 to T31	○	○
F030 to F033	2nd auxiliary function code signals	B00 to B31	○	○
F034.0 to F034.2	Gear selection signals (output)	GR1O,GR2O,GR3O	-	○
F034.3	4th serial spindle ready signals	SRSP4R	○	○
F034.4	3rd serial spindle ready signals	SRSP3R	○	○
F034.5	2nd serial spindle ready signals	SRSP2R	○	○
F034.6	1st serial spindle ready signals	SRSP1R	○	○
F034.7	All-spindle operation ready signal	SRSRDY	○	○
F035.0	Spindle speed fluctuation detection alarm signal	SPAL	○	○

Address	Signal name	Symbol	T series	M series
F036.0 to F037.3	S 12-bit code signals	R01O to R12O	○	○
F038.0	Spindle clamp signal	SCLPA	○	○
F038.1	Spindle unclamp signal	SUCLPA	○	○
F038.2	Spindle enable signal	ENB2	○	○
F038.3		ENB3	○	○
F039.0	Spindle positioning mode signals	MSPOSA	○	○
F039.1	Spindle enable signal	ENB4	○	○
F039.2	Oscillation -in-progress signal	CHPMD	○	○
F039.3	Oscillation cycle signal	CHPCYL	○	○
F040,F041	Actual spindle speed signals	AR00 to AR15	○	○
F043.0 to F043.3	Phase error monitor signal (for each spindle)	SYCAL1 to SYCAL4	○	○
F044.1	Cs contour control change completion signal	FSCSL	○	○
F044.2	Spindle synchronous speed control completion signal	FSPSY	○	○
F044.3	Spindle phase synchronization control completion signal	FSPPH	○	○
F044.4	Phase error monitor signal	SYCAL	○	○
F045.0	Alarm signals (serial spindle)	ALMA	○	○
F045.1	Speed zero signals (serial spindle)	SSTA	○	○
F045.2	Speed detection signals (serial spindle)	SDTA	○	○
F045.3	Spindle speed arrival signal (serial spindle)	SARA	○	○
F045.4	Load detection signals 1(serial spindle)	LDT1A	○	○
F045.5	Load detection signals 2(serial spindle)	LDT2A	○	○
F045.6	Torque limit state signals (serial spindle)	TLMA	○	○
F045.7	Orientation completion signals (serial spindle)	ORARA	○	○
F046.0	Power line switch signals (serial spindle)	CHPA	○	○
F046.1	Spindle switch completion signals (serial spindle)	CFINA	○	○
F046.2	Output switch signals (serial spindle)	RCHPA	○	○
F046.3	Output switch completion signals (serial spindle)	RCFNA	○	○
F046.4	Subordinate operation status signals (serial spindle)	SLVSA	○	○
F046.5	Position coder orientation proximity signal (serial spindle)	PORA2A	○	○
F046.6	Magnetic sensor orientation completion signal (serial spindle)	MORA1A	○	○
F046.7	Magnetic sensor orientation proximity signals (serial spindle)	MORA2A	○	○
F047.0	Position coder one-rotation signal detection status signals (serial spindle)	PC1DTA	○	○
F047.1	Incremental orientation mode signals(serial spindle)	INCSTA	○	○
F047.3	Synchronous orientation enable signal	SORENA	○	○
F047.4	Motor excitation off state signal (serial spindle)	EXOFA	○	○
F047.6	One-rotation signal detection status signal for Cs contour control (serial spindle)	CS1DTA	○	○
F047.7	Reference position establishment completion signal (serial spindle)	CSYFNA	○	○
F048.4	Cs axis origin established state signals	CSPENA	○	○
F049.0	Alarm signals (serial spindle)	ALMB	○	○
F049.1	Speed zero signals (serial spindle)	SSTB	○	○
F049.2	Speed detection signals (serial spindle)	SDTB	○	○
F049.3	Spindle speed arrival signal (serial spindle)	SARB	○	○
F049.4	Load detection signals 1(serial spindle)	LDT1B	○	○
F049.5	Load detection signals 2(serial spindle)	LDT2B	○	○
F049.6	Torque limit state signals (serial spindle)	TLMB	○	○
F049.7	Orientation completion signals (serial spindle)	ORARB	○	○

Address	Signal name	Symbol	T series	M series
F050.0	Power line switch signals (serial spindle)	CHPB	○	○
F050.1	Spindle switch completion signals (serial spindle)	CFINB	○	○
F050.2	Output switch signals (serial spindle)	RCHPB	○	○
F050.3	Output switch completion signals (serial spindle)	RCFNB	○	○
F050.4	Subordinate operation status signals (serial spindle)	SLVSB	○	○
F050.5	Position coder orientation proximity signal (serial spindle)	PORA2B	○	○
F050.6	Magnetic sensor orientation completion signal (serial spindle)	MORA1B	○	○
F050.7	Magnetic sensor orientation proximity signals (serial spindle)	MORA2B	○	○
F051.0	Position coder one-rotation signal detection status signals (serial spindle)	PC1DTB	○	○
F051.1	Incremental orientation mode signals(serial spindle)	INCSTB	○	○
F051.3	Synchronous orientation enable signal	SORENB	○	○
F051.4	Motor excitation off state signal (serial spindle)	EXOFB	○	○
F051.6	One-rotation signal detection status signal for Cs contour control (serial spindle)	CS1DTB	○	○
F051.7	Reference position establishment completion signal (serial spindle)	CSYFNB	○	○
F052.4	Cs axis origin established state signals	CSPENB	○	○
F053.0	Key input disable signal	INHKY	○	○
F053.1	Program screen display mode signal	PRGDPL	○	○
F053.2	Input/output busy signal	IOBSY	○	○
F053.3	Input/output alarm signal	IOALM	○	○
F053.4	Background editing signal	BGEACT	○	○
F053.7	Key code read completion signal	EKENB	○	○
F054,F055	Custom macro output signals	UO000 to UO015	○	○
F056 to F059		UO100 to UO131	○	○
F060.0	Read completion signal for external data input	EREND	○	○
F060.1	Search completion signal for external data input	ESEND	○	○
F060.2	Search cancel signal for external data input	ESCAN	○	○
F061.0	B axis unclamp signal	BUCLP	-	○
F061.1	B axis clamp signal	BCLP	-	○
F061.2	Hard copy cancellation request reception signal	HCAB2	○	○
F061.3	Hard copy execution status signal	HCEXE	○	○
F061.4	Manual tool compensation uncompleted signal	MTLANG	○	-
F061.5	Manual tool compensation completion signal	MTLA	○	-
F062.0	AI contour control mode signal	AICC	○	○
F062.3	Spindle 1 under measurement signal	S1MES	○	-
F062.4	Spindle 2 under measurement signal	S2MES	○	-
F062.6	Three-dimensional coordinate conversion mode signal	D3ROT	○	○
F062.7	Target part count reached signal	PRTSF	○	○
F063.0	Polygon master axis not arrival signal	PSE1	○	○
F063.1	Polygon synchronization axis not arrival signal	PSE2	○	○
F063.2	Polygon spindle speed arrival signal	PSAR	○	○
F063.3	Path spindle command confirmation signal	COSP1	●	●
F063.4		COSP2	●	●
F063.6	Waiting signal	WATO	●	●
F063.7	Polygon synchronization under way signal	PSYN	○	○
F064.0	Tool change signal	TLCH	○	○
F064.1	New tool select signal	TLNW	○	○
F064.2	Individual tool change signal	TLCHI	○	○

Address	Signal name	Symbol	T series	M series
F064.3	Tool life arrival notice signal	TLCHB	○	○
F064.5	Path spindle command confirmation signal	COSP	●	●
F064.6	Path interference check in progress signal	TICHK	●	-
F064.7	Path interference alarm signal	TIALM	●	-
F065.0	Spindle rotation direction signals	RGSPP	○	○
F065.1		RGSPM	○	○
F065.2	Spindle synchronous speed ratio control clamp signal	RSMAX	○	○
F065.4	Retract completion signal	RTRCTF	○	○
F065.6	EGB mode signal	SYNMOD	○	○
F066.1	Rigid tapping retraction completion signal	RTPT	○	○
F066.2	Feed zero signal	FEED0	○	○
F066.5	Small-hole peck drilling cycle in progress signal	PECK2	-	○
F070,F071	Position switch signals	PSW01 to PSW16	○	○
F072	Software operator’s panel general-purpose switch signals	OUT0 to OUT7	○	○
F073.0	Software operator’s panel signal (MD1)	MD1O	○	○
F073.1	Software operator’s panel signal (MD2)	MD2O	○	○
F073.2	Software operator’s panel signal (MD4)	MD4O	○	○
F073.4	Software operator’s panel signal (ZRN)	ZRNO	○	○
F074	Software operator’s panel general-purpose switch signals	OUT8 to OUT15	○	○
F075.2	Software operator’s panel signal (BDT)	BDTO	○	○
F075.3	Software operator’s panel signal (SBK)	SBKO	○	○
F075.4	Software operator’s panel signal (MLK)	MLKO	○	○
F075.5	Software operator’s panel signal (DRN)	DRNO	○	○
F075.6	Software operator’s panel signal (KEY1 to KEY4)	KEYO	○	○
F075.7	Software operator’s panel signal (*SP)	SPO	○	○
F076.0	Software operator’s panel signal (MP1)	MP1O	○	○
F076.1	Software operator’s panel signal (MP2)	MP2O	○	○
F076.3	Rigid tapping-in-progress signal	RTAP	○	○
F076.4	Software operator’s panel signal (ROV1)	ROV1O	○	○
F076.5	Software operator’s panel signal (ROV2)	ROV2O	○	○
F077.0	Software operator’s panel signal (HS1A)	HS1AO	○	○
F077.1	Software operator’s panel signal (HS1B)	HS1BO	○	○
F077.2	Software operator’s panel signal (HS1C)	HS1CO	○	○
F077.3	Software operator’s panel signal (HS1D)	HS1DO	○	○
F077.6	Software operator’s panel signal (RT)	RTO	○	○
F078	Software operator’s panel signal (*FV0 to *FV7)	*FV0O to *FV7O	○	○
F079,F080	Software operator’s panel signal (*JV0 to *JV15)	*JV0O to *JV15O	○	○
F081.0,F081.2, F081.4,F081.6	Software operator’s panel signal (+J1 to +J4)	+J1O to +J4O	○	○
F081.1,F081.3, F081.5,F081.7	Software operator’s panel signal (-J1 to -J4)	-J1O to -J4O	○	○
F082.2	Reverse movement signal	RVSL	-	○
F082.6	EGB synchronization mode confirmation signal	EGBSM	○	○
F084,F085	Output signals for P-code macro	EUO00 to EUO15	○	○
F090.0	Servo axis unexpected disturbance torque detection signal	ABTQSV	○	○
F090.1	1st spindle unexpected disturbance torque detection signal	ABTSP1	○	○
F090.2	2nd spindle unexpected disturbance torque detection signal	ABTSP2	○	○

Address	Signal name	Symbol	T series	M series
F090.3	3rd spindle unexpected disturbance torque detection signal	ABTSP3	○	○
F090.4	Servo motor spindle synchronization mode acceleration/deceleration completion signal	SYAR	○	○
F090.5	Servo motor spindle synchronization mode signal	SYSSM	○	○
F090.6	Servo motor spindle control mode acceleration/deceleration completion signal	SVAR	○	○
F090.7	Servo motor spindle control mode signal	SVSPM	○	○
F091.0	Reverse movement signal	MRVMD	○	○
F091.1	Direction change prohibition signal	MNCHG	○	○
F091.2	Reverse movement prohibition signal	MRVSP	○	○
F091.3	Check mode confirmation signal	MMMOD	○	○
F091.4	4th spindle unexpected disturbance torque detection signal	ABTSP4	○	○
F091.5	Auxiliary function output block reverse movement enable output signal	ADCO	○	○
F092.3	Tool retraction mode signal	TRACT	○	○
F092.4	Tool retraction axis movement signal	TRMTN	○	○
F092.5	Tool return completion signal	TRSPS	○	○
F093.0	Periodic maintenance lifetime warning signal	LIFOVR	○	○
F093.1	Alarm level detection signal	SFAN	○	○
F093.2	Tool life counting disabled signal	LFCIF	○	○
F093.3	Warning level detection signal	WFAN	○	○
F093.4	Servo warning detail signals	SVWRN1	○	○
F093.5		SVWRN2	○	○
F093.6		SVWRN3	○	○
F093.7		SVWRN4	○	○
F094	Reference position return end signals	ZP1 to ZP8	○	○
F096	2nd reference position return completion signals	ZP21 to ZP28	○	○
F098	3rd reference position return completion signals	ZP31 to ZP38	○	○
F100	4th reference position return completion signals	ZP41 to ZP48	○	○
F102	Axis moving signals	MV1 to MV8	○	○
F104	In-position signals	INP1 to INP8	○	○
F106	Axis moving direction signals	MVD1 to MVD8	○	○
F108	Mirror image check signals	MMI1 to MMI8	○	○
F110	Controlled axis detach status signals	MDTCH1 to MDTCH8	○	○
F112	Distribution completion signals (PMC axis control)	EADEN1 to EADEN8	○	○
F114	Torque limit reached signals	TRQL1 to TRQL8	○	○
F118	Synchronous/composite/superimposed control under way signals	SYN1O to SYN8O	○	○
F120	Reference position establishment signals	ZRF1 to ZRF8	○	○
F122	High-speed skip status signals	HDO0 to HDO7	○	○
F124	Overtravel alarm signals	+OT1 to +OT8	○	○
F126		-OT1 to -OT8	○	○
F129.5	Override 0% signal (PMC axis control)	EOV0	○	○
F129.7	Controlled axis selection status signals (PMC axis control)	*EAXSL	○	○
F130.0	In-position signal (PMC axis control)	EINPA	○	○
F130.1	Following zero checking signals (PMC axis control)	ECKZA	○	○
F130.2	Alarm signal (PMC axis control)	EIALA	○	○
F130.3	Auxiliary function executing signals (PMC axis control)	EDENA	○	○

Address	Signal name	Symbol	T series	M series
F130.4	Axis moving signals (PMC axis control)	EGENA	○	○
F130.5	Positive-direction overtravel signals (PMC axis control)	EOTPA	○	○
F130.6	Negative-direction overtravel signals (PMC axis control)	EOTNA	○	○
F130.7	Axis control command read completion signals (PMC axis control)	EBSYA	○	○
F131.0	Auxiliary function strobe signal (PMC axis control)	EMFA	○	○
F131.1	Buffer full signals (PMC axis control)	EABUFA	○	○
F131.2	Auxiliary function 2 strobe signal (PMC axis control)	EMF2A	○	○
F131.3	Auxiliary function 3 strobe signal (PMC axis control)	EMF3A	○	○
F132,F142	Auxiliary function code signals (PMC axis control)	EM11A to EM48A	○	○
F133.0	In-position signal (PMC axis control)	EINPB	○	○
F133.1	Following zero checking signals (PMC axis control)	ECKZB	○	○
F133.2	Alarm signal (PMC axis control)	EIALB	○	○
F133.3	Auxiliary function executing signals (PMC axis control)	EDENB	○	○
F133.4	Axis moving signals (PMC axis control)	EGENB	○	○
F133.5	Positive-direction overtravel signals (PMC axis control)	EOTPB	○	○
F133.6	Negative-direction overtravel signals (PMC axis control)	EOTNB	○	○
F133.7	Axis control command read completion signals (PMC axis control)	EBSYB	○	○
F134.0	Auxiliary function strobe signal (PMC axis control)	EMFB	○	○
F134.1	Buffer full signals (PMC axis control)	EABUFB	○	○
F134.2	Auxiliary function 2 strobe signal (PMC axis control)	EMF2B	○	○
F134.3	Auxiliary function 3 strobe signal (PMC axis control)	EMF3B	○	○
F135,F145	Auxiliary function code signals (PMC axis control)	EM11B to EM48B	○	○
F136.0	In-position signal (PMC axis control)	EINPC	○	○
F136.1	Following zero checking signals (PMC axis control)	ECKZC	○	○
F136.2	Alarm signal (PMC axis control)	EIALC	○	○
F136.3	Auxiliary function executing signals (PMC axis control)	EDENC	○	○
F136.4	Axis moving signals (PMC axis control)	EGENC	○	○
F136.5	Positive-direction overtravel signals (PMC axis control)	EOTPC	○	○
F136.6	Negative-direction overtravel signals (PMC axis control)	EOTNC	○	○
F136.7	Axis control command read completion signals (PMC axis control)	EBSYC	○	○
F137.0	Auxiliary function strobe signal (PMC axis control)	EMFC	○	○
F137.1	Buffer full signals (PMC axis control)	EABUFC	○	○
F137.2	Auxiliary function 2 strobe signal (PMC axis control)	EMF2C	○	○
F137.3	Auxiliary function 3 strobe signal (PMC axis control)	EMF3C	○	○
F138,F148	Auxiliary function code signals (PMC axis control)	EM11C to EM48C	○	○
F139.0	In-position signal (PMC axis control)	EINPD	○	○
F139.1	Following zero checking signals (PMC axis control)	ECKZD	○	○
F139.2	Alarm signal (PMC axis control)	EIALD	○	○
F139.3	Auxiliary function executing signals (PMC axis control)	EDEND	○	○
F139.4	Axis moving signals (PMC axis control)	EGEND	○	○
F139.5	Positive-direction overtravel signals (PMC axis control)	EOTPD	○	○

Address	Signal name	Symbol	T series	M series
F139.6	Negative-direction overtravel signals (PMC axis control)	EOTND	○	○
F139.7	Axis control command read completion signals (PMC axis control)	EBSYD	○	○
F140.0	Auxiliary function strobe signal (PMC axis control)	EMFD	○	○
F140.1	Buffer full signals (PMC axis control)	EABUFD	○	○
F140.2	Auxiliary function 2 strobe signal (PMC axis control)	EMF2D	○	○
F140.3	Auxiliary function 3 strobe signal (PMC axis control)	EMF3D	○	○
F141,F151	Auxiliary function code signals (PMC axis control)	EM11D to EM48D	○	○
F154.0	Number of remaining tools notification signal	TLAL	-	○
F160,F161	Multi-spindle address P signals	MSP00 to MSP15	○	○
F168.0	Alarm signals (serial spindle)	ALMC	○	○
F168.1	Speed zero signals (serial spindle)	SSTC	○	○
F168.2	Speed detection signals (serial spindle)	SDTC	○	○
F168.3	Spindle speed arrival signal (serial spindle)	SARC	○	○
F168.4	Load detection signals 1(serial spindle)	LDT1C	○	○
F168.5	Load detection signals 2(serial spindle)	LDT2C	○	○
F168.6	Torque limit state signals (serial spindle)	TLMC	○	○
F168.7	Orientation completion signals (serial spindle)	ORARC	○	○
F169.0	Power line switch signals (serial spindle)	CHPC	○	○
F169.1	Spindle switch completion signals (serial spindle)	CFINC	○	○
F169.2	Output switch signals (serial spindle)	RCHPC	○	○
F169.3	Output switch completion signals (serial spindle)	RCFNC	○	○
F169.4	Subordinate operation status signals (serial spindle)	SLVSC	○	○
F169.5	Position coder orientation proximity signal (serial spindle)	PORA2C	○	○
F169.6	Magnetic sensor orientation completion signal (serial spindle)	MORA1C	○	○
F169.7	Magnetic sensor orientation proximity signals (serial spindle)	MORA2C	○	○
F170.0	Position coder one-rotation signal detection status signals (serial spindle)	PC1DTC	○	○
F170.1	Incremental orientation mode signals(serial spindle)	INCSTC	○	○
F170.3	Synchronous orientation enable signal	SORENC	○	○
F170.4	Motor excitation off state signal (serial spindle)	EXOFC	○	○
F170.6	One-rotation signal detection status signal for Cs contour control (serial spindle)	CS1DTC	○	○
F170.7	Reference position establishment completion signal (serial spindle)	CSYFNC	○	○
F171.4	Cs axis origin established state signals	CSPENC	○	○
F172.6	Absolute position detector battery voltage zero alarm signal	PBATZ	○	○
F172.7	Absolute position detector battery voltage low alarm signal	PBATL	○	○
F180	Torque limit reach signals for reference point setting with mechanical stopper	CLRCH1 to CLRCH8	○	○
F182	Controlling signals (PMC axis control)	EACNT1 to EACNT8	○	○
F184	Unexpected disturbance torque detection signal	ABDT1 to ABDT8	○	○
F190	Torque control mode signal (PMC axis control)	TRQM1 to TRQM8	○	○
F197.0 to F197.3	Flexible synchronization control mode status signals	MFSYNA,MFSYNB MFSYNC,MFSYND	○	○
F200.0 to F201.3	S 12-bit code signals	R01O2 to R12O2	○	○
F202,F203	Actual spindle speed signals	AR002 to AR152	○	○

Address	Signal name	Symbol	T series	M series
F204.0 to F205.3	S 12-bit code signals	R01O3 to R12O3	○	○
F206,F207	Actual spindle speed signals	AR003 to AR153	○	○
F208	EGB mode confirmation signals	EGBM1 to EGBM8	○	○
F210	Machine coordinate match state output signals	SYNMT1 to SYNMT8	○	○
F211	Synchronization compensation enable state output signals	SYNOF1 to SYNOF8	○	○
F264.0 to F265.0	Spindle warning detail signals 1 to 9	SPWRN1 to SPWRN9	○	○
F266.0	Alarm signals (serial spindle)	ALMD	○	○
F266.1	Speed zero signals (serial spindle)	SSTD	○	○
F266.2	Speed detection signals (serial spindle)	SDTD	○	○
F266.3	Spindle speed arrival signal (serial spindle)	SARD	○	○
F266.4	Load detection signals 1(serial spindle)	LDT1D	○	○
F266.5	Load detection signals 2(serial spindle)	LDT2D	○	○
F266.6	Torque limit state signals (serial spindle)	TLMD	○	○
F266.7	Orientation completion signals (serial spindle)	ORARD	○	○
F267.0	Power line switch signals (serial spindle)	CHPD	○	○
F267.1	Spindle switch completion signals (serial spindle)	CFIND	○	○
F267.2	Output switch signals (serial spindle)	RCHPD	○	○
F267.3	Output switch completion signals (serial spindle)	RCFND	○	○
F267.4	Subordinate operation status signals (serial spindle)	SLVSD	○	○
F267.5	Position coder orientation proximity signal (serial spindle)	PORA2D	○	○
F267.6	Magnetic sensor orientation completion signal (serial spindle)	MORA1D	○	○
F267.7	Magnetic sensor orientation proximity signals (serial spindle)	MORA2D	○	○
F268.0	Position coder one-rotation signal detection status signals (serial spindle)	PC1DTD	○	○
F268.1	Incremental orientation mode signals (serial spindle)	INCSTD	○	○
F268.3	Synchronous orientation enable signal	SOREND	○	○
F268.4	Motor excitation off state signal (serial spindle)	EXOFD	○	○
F268.6	One-rotation signal detection status signal for Cs contour control (serial spindle)	CS1DTD	○	○
F268.7	Reference position establishment completion signal (serial spindle)	CSYFND	○	○
F269.4	Cs axis origin established state signals	CSPEND	○	○
F270.0 to F271.3	S 12-bit code signals	R01O4 to R12O4	○	○
F272,F273	Actual spindle speed signals	AR004 to AR154	○	○
F274.0 to F274.3	Cs contour control change completion signal (for each spindle)	FCSS1 to FCSS4	○	○
F274.4 to F274.7	Cs axis coordinate establishment alarm signals (for each spindle)	CSFO1 to CSFO4	○	○
F276 to F277	Custom macro output signals	UO016 to UO031	○	○
F280 to F283		UO200 to UO231	○	○
F284 to F287		UO300 to UO331	○	○
F288.0 to F288.3	Spindle synchronous speed control completion signal (for each spindle)	FSPSY1 to FSPSY4	○	○
F289.0 to F289.3	Spindle phase synchronization control completion signal (for each spindle)	FSPPH1 to FSPPH4	○	○
F290.2	DeviceNet communication normal signal	DNTCM	○	○
F290.4	High speed program check saving data signal	PCKSV	○	○
F290.5	High speed program check mode signal	PRGMD	○	○

Address	Signal name	Symbol	T series	M series
F293,F294	High-speed position switch signals	HPS01 to HPS16	O	O
F295.6	Dual display forcible end status signal	C2SENO	O	O
F295.7	Key control selection status signal	CNCKYO	O	O
F296	Diameter/radius specification switching in-progress signals (each axis)	DM1 to DM8	O	O
F297.0	Block cancel acknowledgement signal	MBCAN	O	O
F298	Trouble forecast signals for thermal simulation	TDSML1 to TDSML8	O	O
F299	Trouble forecast signals for disturbance level	TDFTR1 to TDFTR8	O	O
F306.4	DC-link failure detection state signal (serial spindle)	VDCABA	O	O
F307.0	Adaptive resonance elimination filter search completion signal (serial spindle)	FRDTEA	O	O
F307.1	Power failure detection signal (serial spindle)	XPFLA	O	O
F307.2	Inertia estimation completion signal (serial spindle)	INESFNA	O	O
F308.4	DC-link failure detection state signal (serial spindle)	VDCABB	O	O
F309.0	Adaptive resonance elimination filter search completion signal (serial spindle)	FRDTEB	O	O
F309.1	Power failure detection signal (serial spindle)	XPFLB	O	O
F309.2	Inertia estimation completion signal (serial spindle)	INESFNB	O	O
F310.4	DC-link failure detection state signal (serial spindle)	VDCABC	O	O
F311.0	Adaptive resonance elimination filter search completion signal (serial spindle)	FRDTEC	O	O
F311.1	Power failure detection signal (serial spindle)	XPFLC	O	O
F311.2	Inertia estimation completion signal (serial spindle)	INESFNC	O	O
F312.4	DC-link failure detection state signal (serial spindle)	VDCABD	O	O
F313.0	Adaptive resonance elimination filter search completion signal (serial spindle)	FRDTED	O	O
F313.1	Power failure detection signal (serial spindle)	XPFLD	O	O
F313.2	Inertia estimation completion signal (serial spindle)	INESFND	O	O
F315.0	Tool skip completion signal	TLSKF	O	O
F315.1	Tool search in-progress signal	TLMSRH	O	O
F315.2	Tool management data modification in-progress signal	TLMG10	O	O
F315.4	Tool management data output in-progress signal	TLMOT	O	O
F315.6	Life expiration signal	TMFNFD	O	O
F315.7	Tool management data edit in-progress signal	TLMEM	O	O
F316.6	Program restart MDI program output completion signal	SQMPR	O	O
F316.7	Program restart MDI program execution completion signal	SQMPE	O	O
F328.0 to F328.3	Tool change signals 1 to 4	TLCH1 to TLCH4	O	O
F328.4 to F328.7	Individual tool change signals 1 to 4	TLCHI1 to TLCHI4	O	O
F329.0 to F329.3	Tool skip completion signals 1 to 4	TLSKF1 to TLSKF4	O	O
F329.4 to F329.7	Tool life expiration notice signals 1 to 4	TLCHB1 to TLCHB4	O	O
F341	Synchronous master axis confirmation signals	SYCM1 to SYCM8	O	O
F342	Synchronous slave axis confirmation signals	SYCS1 to SYCS8	O	O
F343	Composite axis confirmation signals	MIXO1 to MIXO8	O	O
F344	Superimposed control master axis confirmation signals	OVMO1 to OVMO8	O	O
F345	Superimposed control slave axis confirmation signals	OVSO1 to OVSO8	O	O
F346	Parking axis confirmation signals	SMPK1 to SMPK8	O	O
F347.7	3-dimensional coordinate conversion manual interrupt mode in-progress signal	D3MI	O	O

Address	Signal name	Symbol	T series	M series
F351.0 to F351.3	Simple spindle EGB mode signals	SSEGBM1 to SSEGBM4	○	○
F358	Each axis workpiece coordinate system preset completion signals	WPSF1 to WPSF8	○	○
F376	Speed zero signals	SVSST1 to SVSST8	○	○
F377	Speed arrival signals	SVSAR1 to SVSAR8	○	○
F381.0 to F381.3	Flexible synchronization control phase synchronization end signals	PHFINA to PHFIND	○	○
F400.1	Spindle unclamp signal	SUCLPB	○	○
F400.2		SUCLPC	○	○
F400.3		SUCLPD	○	○
F401.1	Spindle clamp signal	SCLPB	○	○
F401.2		SCLPC	○	○
F401.3		SCLPD	○	○
F402.1	Spindle positioning mode signals	MSPOSB	○	○
F402.2		MSPOSC	○	○
F402.3		MSPOSD	○	○
F403.0	Signal for indicating a positional deviation error alarm for axis synchronous control	SYNER	○	○
F404.0	Path spindle command confirmation signal	COSP3	●	●
F404.1		COSP4	●	●
F512.0	Macro call executing signal	MCEXE	○	○
F512.1	Mode change request signal	MCRQ	○	○
F512.2	Abnormal end signal	MCSP	○	○
F513.0	Mode notification signal	MD1R	○	○
F513.1		MD2R	○	○
F513.2		MD4R	○	○
F513.5		DNCIR	○	○
F513.7		ZRNR	○	○
F514,F515	Call program confirmation signal	MCEX1 to MCEX16	○	○
F517.0 to F517.7	Reference position match signals	RP11 to RP18	○	○
F518.0 to F518.7	2nd reference position match signals	RP21 to RP28	○	○
F520.0	Automatic data backup executing signal	ATBK	○	○
F520.3	In-acceleration/deceleration signal	ACDEC	-	○
F521	SV speed control mode in-progress signal	SVREV1 to SVREV8	○	○
F522	Spindle indexing signals for each axis	SPP1 to SPP8	○	○
F526.5	Dwell status signal	DWL	○	○
F527.6	Phase synchronization for Servo/Spindle synchronous finished signal	SYPFN	○	○
F527.7	Phase synchronization for Servo/Spindle synchronous error signal	SYPER	○	○
F531.6	External device program execution signal	DVCPR	○	○
F531.7	β ready signal	IOLBR	○	○
F532	Axis synchronous control status signals	SYNO1 to SYNO8	○	○
F534.1	Quick program restart under way signal	SRNEX	○	○
F534.4	Middle block start signal	MBSO	○	○
F535.0	I/O Link 1 retry abnormality warning signal	WIOCH1	○	○
F535.1	I/O Link 2 retry abnormality warning signal	WIOCH2	○	○
F535.2	I/O Link 3 retry abnormality warning signal	WIOCH3	○	○
F535.3	SRAM ECC abnormality warning signal	WECCS	○	○
F535.4	Embedded Ethernet communication abnormality warning signal	WETE	○	○

Address	Signal name	Symbol	T series	M series
F535.5	Fast Ethernet communication abnormality warning signal	WETF	○	○
F535.6	FL-net1 communication abnormality warning signal	WFLN1	○	○
F535.7	FL-net2 communication abnormality warning signal	WFLN2	○	○
F536.2	Removal completion signal	RMVED	○	○
F536.3	Assignment completion signal	ASNED	○	○
F536.4	Exchange completion signal	EXCED	○	○
F536.7	Initial axis assignment signal	INIST	○	○
F545.0	Display language switch completion signal	FLANG	○	○
F545.1	Inter-path flexible synchronization mode signal Advanced superimposition mode signal	OVLNS	○	○
F545.4	DeviceNet communication abnormal signal	DNTER	○	○
F546.0 to F546.3	Cs contour control high speed switching completion signals	CSMC1 to CSMC4	○	○
F546.4	Groove of thread measurement completion signal	GTMC	○	○
F546.5	Groove of thread measurement error signal	GTME	○	○
F547.5	Programs not saved status signal	HPMNTS	○	○
F547.6	Programs saving in progress signal	HPMSVM	○	○
F547.7	Programs save error signal	HPMERR	○	○
F553.0 to F553.3	Automatic phase synchronization error detection signals	PHERA, PHERB, PHERC, PHERD	○	○
F553.4 to F553.7	Flexible synchronization control mode selecting signals	FSYSA, FSYSB, FSYSC, FSYSD	○	○
F558.0	Notification signal for modification of C Language Executor program	CDCEX	○	○
F558.1	Notification signal for modification of 1st path PMC Ladder program	CDLAD1	○	○
F558.2	Notification signal for modification of 2nd path PMC Ladder program	CDLAD2	○	○
F558.3	Notification signal for modification of 3rd path PMC Ladder program	CDLAD3	○	○
F558.4	Notification signal for modification of Dual Check Safety PMC Ladder program	CDDCL	○	○
F558.5	Notification signal for modification of CNC parameter	CDPRM	○	○
F558.6	Notification signal for modification of 4th path PMC Ladder program	CDLAD4	○	○
F558.7	Notification signal for modification of 5th path PMC Ladder program	CDLAD5	○	○
F559	Excess synchronization error signals	SEO1 to SEO8	○	○
F564 to F567	3rd M function code signals	M300 to M331	○	○
F568 to F571	4th M function code signals	M400 to M431	○	○
F572 to F575	5th M function code signals	M500 to M531	○	○
F577.0 to F577.3	Spindle position save completion signal	SPMFN1 to SPMFN4	○	○
F577.4 to F577.7	Spindle position save error signal	SPMER1 to SPMER4	○	○
F578.2	Web browser connection status signal	WBCNT	○	○
F578.5	NC data output signal	ALLO	○	○
F580 to F583	Extended actual spindle speed signals	ARE00 to ARE31	○	○
F584 to F587		ARE002 to ARE312	○	○
F588 to F591		ARE003 to ARE313	○	○
F592 to F595		ARE004 to ARE314	○	○

Address	Signal name	Symbol	T series	M series
F598	Stored stroke limit range switching confirmation signals	+OT11O, -OT11O, +OT12O, -OT12O, +OT2O, -OT2O, +OT3O, -OT3O	○	○
F599.0	Stored stroke limit range switching finish signal	OTSWFN	○	○
F599.3	Servo loop gain / in-position width switching confirmation signal	GISO	○	○
F599.4 to F599.5	Time constant of acceleration / deceleration after interpolation for cutting feed switching confirmation signals	CTC2O to CTC3O	○	○
F599.6 to F599.7	Time constant of acceleration / deceleration after interpolation for rapid traverse switching confirmation signals	RTC2O to RTC3O	○	○
F687	Dual control axes status signal	SVMWS1 to SVMWS8	○	○
F708 to F711	S32-bit code signals	RE01O to RE32O	○	○
F712 to F715		RE01O2 to RE32O2	○	○
F716 to F719		RE01O3 to RE32O3	○	○
F720 to F723		RE01O4 to RE32O4	○	○
F730	Target distance attainment status signal	TDA1 to TDA8	○	○
F747	DO signal for Data transfer between PMC and DCSPMC	TDCF00 to TDCF07	○	○

B LIST OF FUNCTIONS USING PMC SIGNALS OTHER THAN G/F ADDRESS

B.1 LIST OF FUNCTIONS USING PMC SIGNALS OTHER THAN G/F ADDRESS

Functions using PMC signals other than G/F address in addition to the function to create applications and the communication function that are listed in “SAFETY PRECAUTIONS” are as follows.

Function name	Signal types	Access types to signal	Related parameter numbers	Reference item
High-Speed Position Switch	Y	Write	HPF(No.8501#0) No.8565	1.2.11 High-Speed Position Switch
Flexible Path Axis Assignment	R	Write	No.11554	1.18.1 Outputting States of Individual Axes
Stored Stroke Limit Range Switching Function by Signal	D	Read	No.1313	2.3.10 Stored Stroke Limit Range Switching Function by Signal
Manual Linear/Circular Interpolation	R	Read Write	MRI(No.7106#3) MRO(No.7106#4) No.13541 No.13542	3.4 Manual Linear/Circular Interpolation
Manual Reference Position Return	X	Read	GDC(No.3006#0) XSG(No.3008#2) No.3013 No.3014	4.1 Manual Reference Position Return
Spindle Speed Command Clamp	R	Read	No.3773	11.26 Spindle Speed Command Clamp
Custom Macro	R	Write	IFR(No.6020#2) No.6094	13.6 Custom Macro
Energy Saving Level Selecting Function	X,Y,A,R, T,K,C,D	Read	Set signals on the display	14.1.25 Energy Saving Level Selecting Function
Automatic Tool Length Measurement (M Series) /Automatic Tool Offset (T Series)	X	Read	XSG(No.3008#2) No.3019	16.2 Automatic Tool Length Measurement (M Series) /Automatic Tool Offset (T Series)
Skip Function	X	Read	XSG(No.3008#2) No.3012	16.3.1 Skip Function
Multi-Step Skip	X	Read	XSG(No.3008#2) No.3012	16.3.5 Multi-Step Skip
Direct Input of Offset Value Measured B (for Lathe System)	X	Read	XSG(No.3008#2) No.3019	16.4.2 Direct Input of Offset Value Measured B (for Lathe System)
PMC Axis Control	X	Read	XSG(No.3008#2) No.3019 SKE(No.8001#7)	17.1.1 PMC Axis Control
Extended External Machine Zero Point Shift	R	Read	EMS(No.1203#0) No.1280	17.3 Extended External Machine Zero Point Shift

Function name	Signal types	Access types to signal	Related parameter numbers	Reference item
Communication Retry Monitoring Function	R9051 (Z51) R9057 (Z57) R9165 (Z165)	Read	Fixed signals	19.3.2 Communication Retry Monitoring Function

C LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE PROGRAM COMMAND

C.1 LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF G CODE

Functions including address P in the argument of G code are shown below.

Function name	Machining center system	Lathe system			Reference item
		G code system			
		A	B	C	
Dwell	G04	G04	G04	G04	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
G code preventing buffering	G04.1	G04.1	G04.1	G04.1	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
AI contour control II(high-precision contour control/AI high-precision contour control compatible command)	G05	G05	G05	G05	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
AI contour control (advanced preview control compatible command)	G08	G08	G08	G08	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Programmable data input	G10	G10	G10	G10	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Programmable parameter input	G10	G10	G10	G10	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Programmable internal data change	G10.8	G10.8	G10.8	G10.8	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES)
Machine configuration selecting function	G10.8	G10.8	G10.8	G10.8	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Spindle speed fluctuation detection	G26	G26	G26	G26	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Reference position return	G30	G30	G30	G30	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Multi-step skip	G31	G31	G31	G31	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Torque limit skip	G31	G31	G31	G31	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Continuous high-speed skip	G31	G31	G31	G31	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Skip function for EGB axis	G31.8	G31.8	G31.8	G31.8	CONNECTION MANUAL(FUNCTION)
Wheel wear compensation	G41	G41	G41	G41	OPERATOR'S MANUAL(For Machining Center System) II. PROGRAMMING

Function name	Machining center system	Lathe system G code system			Reference item
		A	B	C	
Scaling	G51	-	G51	G51	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Polygon turning	G51.2	G51.2 (G251)	G51.2 (G251)	G51.2 (G251)	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Synchronous, Composite and Superimposed control by program command	G51.4, G51.5, G51.6, G50.5	G51.4, G51.5, G51.6, G50.5	G51.4, G51.5, G51.6, G50.5	G51.4, G51.5, G51.6, G50.5	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Flexible path axis assignment	G52.1, G52.2, G52.3	G52.1, G52.2, G52.3	G52.1, G52.2, G52.3	G52.1, G52.2, G52.3	CONNECTION MANUAL(FUNCTION)
High-speed G53 function	G53	G53	G53	G53	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Workpiece coordinate system	G54 (G54.1)	G54 (G54.1)	G54 (G54.1)	G54 (G54.1)	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES)
Custom macro	G65, G66, G66.1	G65, G66, G66.1	G65, G66, G66.1	G65, G66, G66.1	OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Execution macro Note1)	G65, G66, G66.1 Note1)	G65, G66, G66.1 Note1)	G65, G66, G66.1 Note1)	G65, G66, G66.1 Note1)	Macro Executor PROGRAMMING MANUAL
Pattern Data Input	G65	G65	G65	G65	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Balanced cutting	-	G68	G68	G68	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(For Lathe System)
Tilted working plane indexing	G68.2	G68.2	G68.2	G68.2	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Tilted working plane indexing by tool axis direction	G68.3,	G68.3,	G68.3,	G68.3,	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES)
Tilted working plane indexing (incremental multi-command)	G68.4	G68.4	G68.4	G68.4	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Multiple repetitive cycles	G70.7 to G76.7	G70 to G76	G70 to G76	G72 to G78	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(For Lathe System) OPERATOR'S MANUAL(For Machining Center System)
Figure copying	G72.1, G72.2	G72.1, G72.2	G72.1, G72.2	G72.1, G72.2	OPERATOR'S MANUAL(COMMON TO T/M SERIES)
Canned cycle	G74, G76	-	-	-	OPERATOR'S MANUAL(For Machining Center System)

Function name	Machining center system	Lathe system G code system A	B	C	Reference item
Canned cycle for grinding	G75, G77, G78, G79	G72, G74	G72, G74	G73, G75	OPERATOR'S MANUAL(For Lathe System) II. PROGRAMMING
Canned cycle	G82 to G84, G87 to G89	G82 to G85, G87 to G89, G83.5, G83.6, G87.5, G87.6	G82 to G85, G87 to G89, G83.5, G83.6, G87.5, G87.6	G82 to G85, G87 to G89, G83.5, G83.6, G87.5, G87.6	OPERATOR'S MANUAL(For Lathe System) OPERATOR'S MANUAL(For Machining Center System)
Electronic gear box	G81	G81	G81	G81	CONNECTION MANUAL(FUNCTION)
Electronic gear box 2 pair	G81.5	G81.5	G81.5	G81.5	CONNECTION MANUAL(FUNCTION)
High-speed peck drilling cycle	-	G83.1	G83.1	G83.1	OPERATOR'S MANUAL(For Lathe System)
Rigid tapping cycle (FS15-T format)	G84.2	G84.2	G84.2	G84.2	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(For Lathe System) OPERATOR'S MANUAL(For Machining Center System)
Left-handed rigid tapping cycle(FS15-T format)	G84.3	-	-	-	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(For Machining Center System)
Constant surface speed control	G96	G96	G96	G96	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Spindle Indexing Function	G96.1 to G96.3	G96.1 to G96.3	G96.1 to G96.3	G96.1 to G96.3	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Spindle control with servo motor	G96.4	G96.4	G96.4	G96.4	CONNECTION MANUAL(FUNCTION) OPERATOR'S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING

Note 1) : Arbitrary G code is optional with the following compilation parameter.
No.9013 to No.9022,No.9034,No.9045 to No.9047,No.9129 to No.9137

C.2 LIST OF FUNCTIONS INCLUDE ADDRESS P IN THE ARGUMENT OF M AND S CODE

Functions including address P in the argument of M and S code are shown below.

Function name	M code format	related parameters	Reference item
Path table operation	M_(P_)Q_	No.11100	CONNECTION MANUAL(FUNCTION)
Waiting M codes	M_P_	No.8110,No.8111, MWP(No.8103#1)	CONNECTION MANUAL(FUNCTION) OPERATOR’S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Waiting M codes of high-speed Type	M_P_	No.8114,No.8115, MWP(No.8103#1)	CONNECTION MANUAL(FUNCTION) OPERATOR’S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
Waiting function by specifying start point	M_P_L_IP	STW(No.8101#1), No.8110,No.8111, MWP(No.8103#1)	CONNECTION MANUAL(FUNCTION) OPERATOR’S MANUAL(COMMON TO T/M SERIES) II. PROGRAMMING
(Custom macro) Subprogram Call	M98P_		OPERATOR’S MANUAL(COMMON to T/M series) II. PROGRAMMING
Program end	M99P_		
External subprogram call	M198P_	SBP(No.3404#2)	OPERATOR’S MANUAL(COMMON to T/M series) III. OPERATION
Custom macro Macro Call Using an M code (include specification of multiple definitions and special macro call)	M_P_	MAA(No.6009#2)	OPERATOR’S MANUAL(COMMON to T/M series) II. PROGRAMMING
Execution macro Subprogram call	M98P_		Macro Executor PROGRAMMING MANUAL
Subprogram call for user program	M_P_		
Execution macro Macro Call Using an M code (include specification of multiple definitions and special macro call)	M_P_		
Multi-spindle	S_P_	MPP(No.3703#3), No.3781	CONNECTION MANUAL(FUNCTION)

D PARAMETER TUNING FOR ENHANCEMENT OF MACHINING QUALITY

This chapter is composed of the following item.
CNC and Servo control parameters setting/tuning are executed as the following sequence.

CNC and Servo control parameters tuning

1 Initialization of parameters

- Setting of recommended values

2 Tuning if there is problem

- 2-1 Tuning of AI contour control — Tuning highly effective parameters for problem
- 2-2 Countermeasure of vibration for lower frequency — Reducing the vibration less than 50Hz

Supplement

Supplement A Details of AI contour control

Supplement B Servo tuning for higher response

D.1 CNC AND SERVO CONTROL PARAMETERS TUNING

1 Initialization of parameters

Set recommended value in CNC and Servo control parameters.

Setting item	Parameter number	Recommended value for BT30			Recommended value for BT40			Recommended value for BT50		
		Fast	Normal	Fine	Fast	Normal	Fine	Fast	Normal	Fine
AI contour control during automatic operation	No.1604#0	1	1	1	1	1	1	1	1	1
Acceleration rate for Acc./Dec. before interpolation [mm/sec^2]	No.1660	4000	2000	1600	2800	800	400	1580	620	300
Time constant for bell-type Acc./Dec. before interpolation [msec]	No.1772	20	32	40	20	32	40	48	60	64
Type for Acc./Dec. after interpolation (Bell-type)	No.1602#6,#3	1	1	1	1	1	1	1	1	1
Time constant for Acc./Dec. after interpolation [msec]	No.1769	24	16	16	24	16	16	24	24	24
Corner feed rate difference [mm/min]	No.1783	500	500	400	500	300	150	632	248	120
Allowable acceleration rate for circular interpolation Allowable acceleration rate for small line blocks [mm/sec^2]	No.1735 No.1737	3000	1500	1200	2100	600	300	1185	465	225
Minimum feed rate for deceleration based on allowable acceleration rate for circular interpolation Minimum feed rate for deceleration based on allowable acceleration rate for small line blocks [mm/min]	No.1732 No.1738	100	100	100	100	100	100	100	100	100
Enable Smooth feedrate control	No.19503#0	1	1	1	1	1	1	1	1	1
Disable acceleration control by block length	No.19517#1,#0	1	1	1	1	1	1	1	1	1
Time constant for 2nd Acc./Dec. after interpolation in cutting feed [msec]	No.11238	0	0	0	0	0	0	0	0	0
Enable Smart overlap function	No.11236#1	1	1	1	1	1	1	1	1	1
Time constant for Acc./Dec. after interpolation of Smart overlap [msec]	No.11248	16	16	16	16	16	16	16	16	16

Setting item	Parameter number	Recommended value for BT30			Recommended value for BT40			Recommended value for BT50		
		Fast	Normal	Fine	Fast	Normal	Fine	Fast	Normal	Fine
Maximum overlap value of Smart overlap (Rapid traverse - Rapid traverse) [Detection unit 1μm/0.1μm]	No.10740	1000/ 10000	1000/ 10000	1000/ 10000	1000/ 10000	1000/ 10000	1000/ 10000	1000/ 10000	1000/ 10000	1000/ 10000
Maximum overlap value of Smart overlap (Rapid traverse – Cutting feed) [Detection unit 1μm/0.1μm]	No.10741	100/ 1000	100/ 1000	100/ 1000	100/ 1000	100/ 1000	100/ 1000	100/ 1000	100/ 1000	100/ 1000
Maximum overlap value of Smart overlap (Cutting feed - Rapid traverse) [Detection unit 1μm/0.1μm]	No.10742	100/ 1000	100/ 1000	100/ 1000	100/ 1000	100/ 1000	100/ 1000	100/ 1000	100/ 1000	100/ 1000

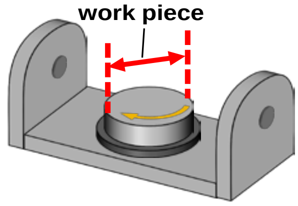
- For large machines such as double colum machines with BT50, use fine setting value as normal setting value.

Setting of rotary axis

Setting value of rotary axis basically should be the same as linear axis. In the case of machining work piece that diameter is mainly 120mm or less on rotary table, decide parameters on the following table with K obtained by the following formula. It reduces machining time.

$$K = \frac{120}{\text{Diameter of work piece[mm]}}$$

$$\textit{Setting value of rotary axis} = (\textit{Recommneded value of linear axis}) \times K$$



- Set K to 3, if K is more than 3.

Recommended setting value of linear axis

Setting item	Parameter number	Recommended value for BT30			Recommended value for BT40			Recommended value for BT50		
		Fast	Normal	Fine	Fast	Normal	Fine	Fast	Normal	Fine
Acceleration rate for Acc./Dec. before interpolation [mm/sec^2]	No.1660	4000	2000	1600	2800	800	400	1580	620	300
Corner feed rate difference [mm/min]	No.1783	500	500	400	500	300	150	632	248	120
Allowable acceleration rate for circular interpolation Allowable acceleration rate for small line blocks [mm/sec^2]	No.1735 No.1737	3000	1500	1200	2100	600	300	1185	465	225

Jerk control (M series)

Setting item	Parameter number	Recommended value
Allowable acceleration change amount [mm/sec^2]	No.1788	0
Allowable acceleration change amount (linear interpolation) [mm/sec^2]	No.1789	0
Ratio of change time of the rate of change of acceleration [%]	No.1790	0

- If machine vibration and machining error cannot be reduced enough by only setting allowable acceleration rate, use Jerk control.

Servo control

Setting item	Parameter number	Recommended value
Velocity loop PI function	No.2003#3	1
1-ms velocity feedback acquisition	No.2006#4	1
HRV3 current control	No.2013#0	1
Variable proportional gain in the stop sate	No.2016#3	1
Velocity loop high cycle management function	No.2017#7	1
Load Inertia ratio (Velocity Loop Gain)	No.2021	256 (αi series motor) 0 (βi series motor)
TCMD filter	No.2067	0
Velocity loop gain override at cutting feed	No.2107	150
Judgment level for stop state for variable proportional gain function in the stop state (specified in detection units)	No.2119	20 (Detection unit 0.1µm)
Servo loop gain in rapid feed (Position gain)	No.2178	3000 (Semi-closed system) 3000 (Full-closed system)
Cutting/rapid feed velocity loop gain variable	No.2202#1	1
Current loop 1/2 PI control function	No.2203#2	0
Cutting/rapid feed position loop gain function	No.2213#6	1
High-speed HRV current control in cutting feed	No.2283#0	1
Current gain magnification in HRV3 mode	No.2334	150
Velocity gain magnification in HRV3 mode	No.2335	200
Servo loop gain (Position gain)	No.1825	5000 (Semi-closed system) 3000 (Full-closed system)
Feed-forward control in rapid feed	No.1800#3	1
Current control setting	No.2004	00000011
Feed-forward	No.2005#1	1
Velocity feed-forward coefficient for rapid feed	No.2069	100
Feed-forward coefficient for rapid feed	No.2092	9800
Feed-forward coefficient for cutting feed	No.2144	10000
Velocity feed-forward coefficient for cutting feed	No.2145	100
Cutting/rapid feed-forward switching function	No.2214#4	1
Default value of the feed-forward timing adjustment parameter	No.2415#1	1
Acceleration monitor function for backlash acceleration	No.2423#7	1
Acceleration monitor time	No.2184	100
Backlash acceleration amount override	No.2296#7	1
Backlash acceleration stop	No.2296#6	1

2 Tuning if there is problem

2-1 Tuning of AI contour control

Tune the parameter according to priority for each purpose (Reducing vibration, Reducing machining error, Enhancing machining quality, Reducing cycle time).

Tuning priority Priority High : ◊◊ / Middle : ◊ / Low : None

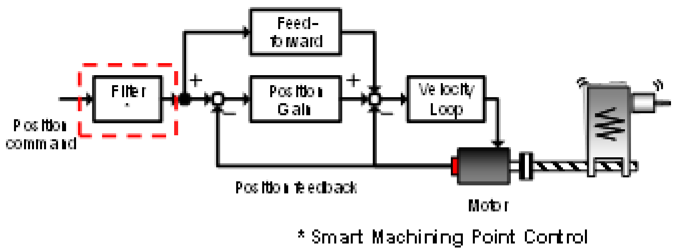
Purpose	Machining	Tuning item (Upper : Parameter number / Lower : Tuning rage)					
		Acc. rate for Acc./Dec. before interpolation (No.1660)	Time const. for bell-type Acc./Dec. before interpolation (No.1772)	Time const. for Acc./Dec. after interpolation (No.1769)	Corner feed rate difference (No.1783)	Allowable acc. rate for circular interpolation (No.1735)	Allowable acc. rate for small line blocks (No.1737)
	Parts	250 to 5000	16 to 64	8 to 48	150 to 1000	200 to 3750	200 to 3750
	Die-mold			8 to24			
Reducing vibration	Parts	Decrease ◊	Increase ◊	Increase ◊◊	Decrease ◊	Decrease	Decrease
	Die-mold			Increase			
Reducing machining error	Parts	Decrease	Increase	Decrease ◊◊	Decrease ◊	Decrease ◊	Decrease
	Die-mold				Decrease	Decrease	Decrease ◊
Enhancing machining quality	Parts	Decrease	Increase	Increase ◊◊	Decrease	Decrease	Decrease
	Die-mold		Increase ◊	Decrease ◊◊			Decrease ◊
Reducing cycle time	Parts	Increase ◊	Decrease ◊◊	Decrease	Increase ◊	Increase	Increase
	Die-mold						

- In case that the priority of parameter is same, tune the parameter with high effect after confirming the effect of each parameter.

2-2 Countermeasure of vibration for lower frequency

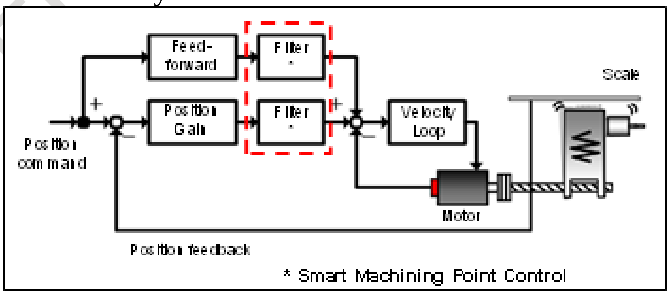
Avoidance of machining point vibration (below 50Hz) after Acc./Dec. by Smart machining point control.

Semi-closed system



Parameter number	Recommended value
No.2564#1	1
No.2638[0.1Hz]	Vibration freq.[Hz]×10
No.2639	10

Full-closed system

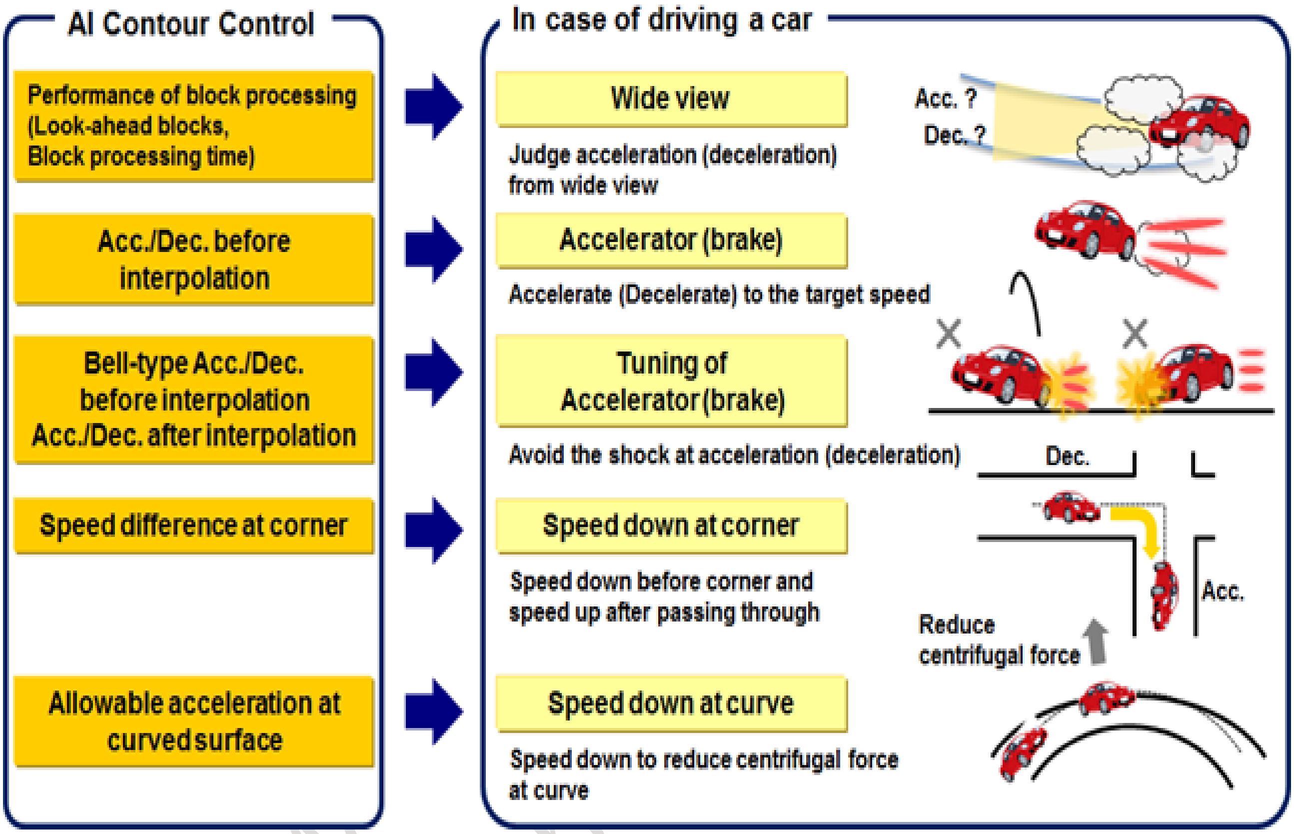


Parameter number	Recommended value
No.2570#4	1
No.2571#3	1
No.2647[0.1Hz]	Vibration freq.[Hz]×10
No.2648	30
No.2649	2510

D.2 SUPPLEMENT

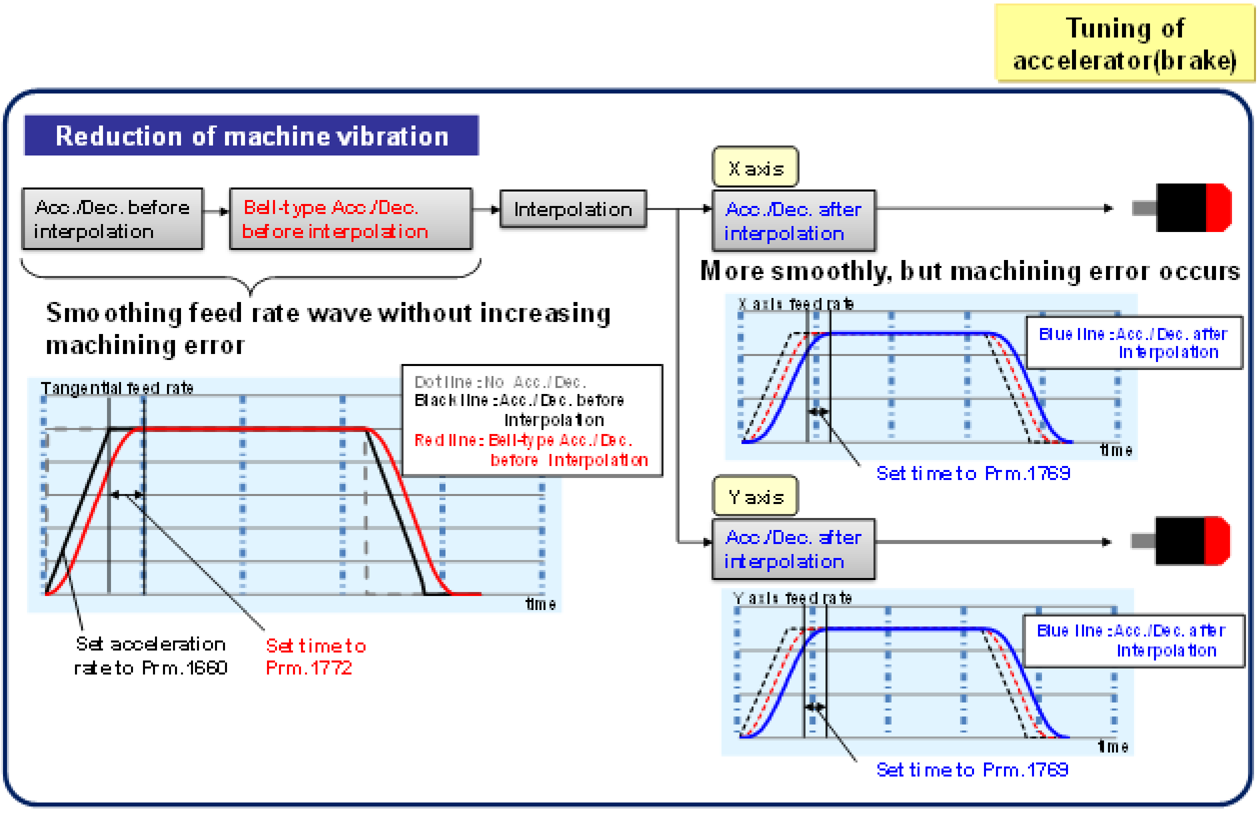
Supplement A. Details of AI contour control

AI contour control is the function to possible high speed and high precision machining with low machine vibration and low machining error by following four functions.
The following four functions can be easily understood as the comparison to driving a car.



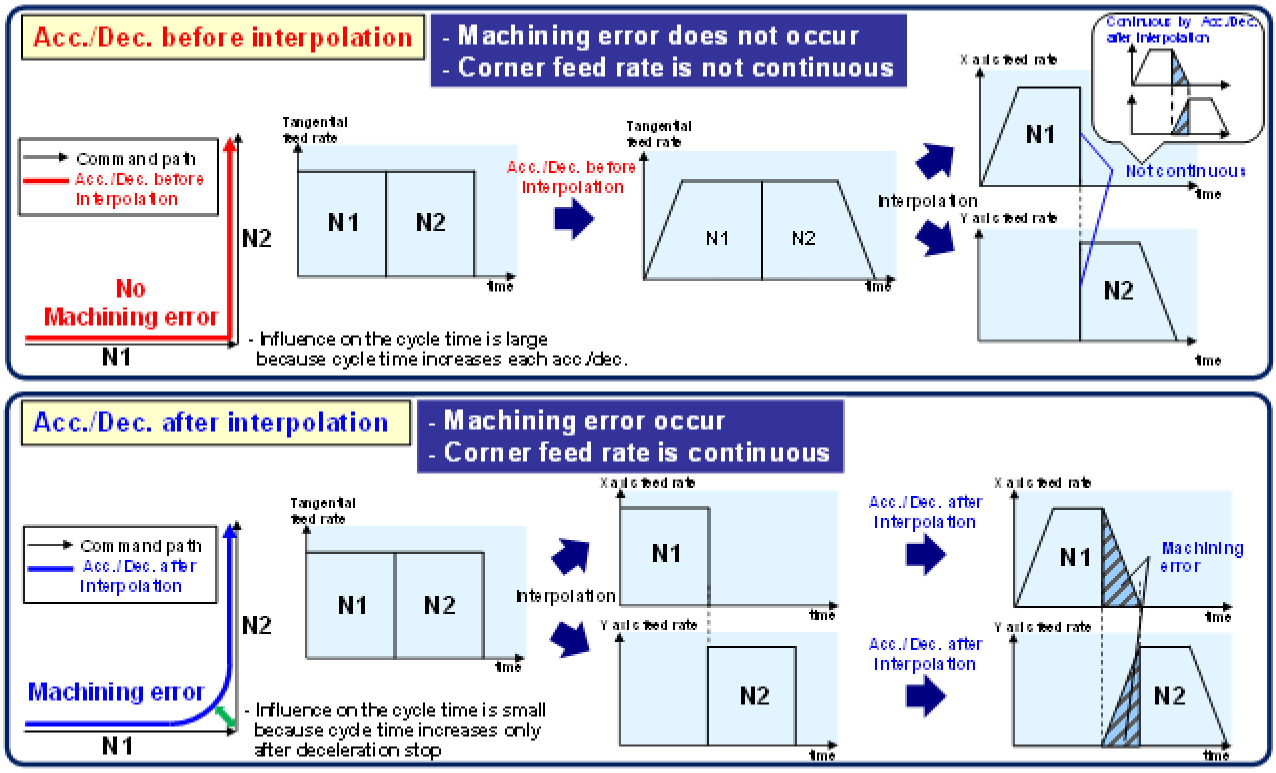
Acc./Dec. before interpolation, Bell-type Acc./Dec. before interpolation, Acc./Dec. after interpolation

Acc./Dec. before interpolation is function to accelerate or decelerate for command feed-rate before CNC interpolates machining program command. Acc./Dec. after interpolation is function to accelerate or decelerate for each axis feed-rate after CNC interpolates.
In Acc./Dec. before interpolation, set acceleration rate (No.1660), and it is possible to produce smooth feed-rate curve by bell-type Acc./Dec. before interpolation (No.1772).
Acc./Dec. before interpolation achieve Acc./Dec. without machining error because CNC interpolates smoothed feed-rate curve with Acc./Dec. before interpolation.
It is possible to more smooth feed-rate curve by using Acc./Dec. after interpolation (No.1769), but Acc./Dec. after interpolation makes machining error because it makes after CNC interpolation.



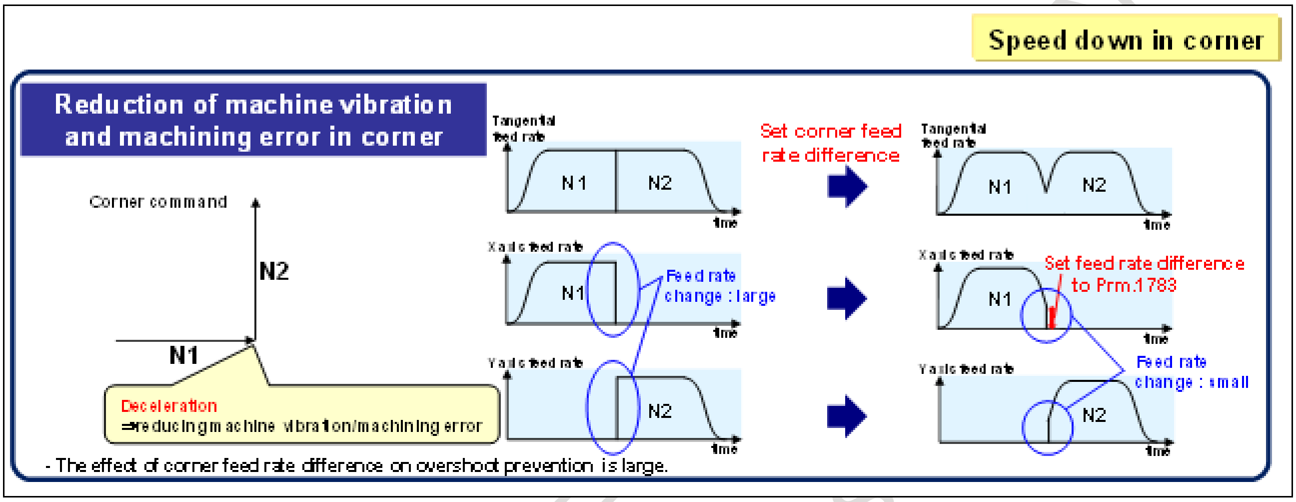
Difference between Acc./Dec. before interpolation and Acc./Dec. after interpolation

Acc./Dec. before interpolation achieve Acc./Dec. without machining error, but feed-rate of each axis is not continuous in the corner and it might cause machine vibration or shock.
It is possible to smooth the discontinuous feed-rate point by using Acc./Dec. after interpolation that accelerate or decelerate for each axis feed-rate after CNC interpolates, but the smoothing of feed-rate by Acc./Dec. after interpolation occurs machining error as described above.
So, both Acc./Dec. before interpolation and Acc./Dec. after interpolation are necessary functions and it is important to use well-balance.

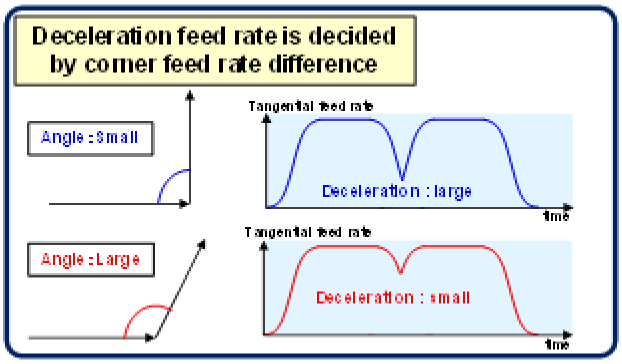


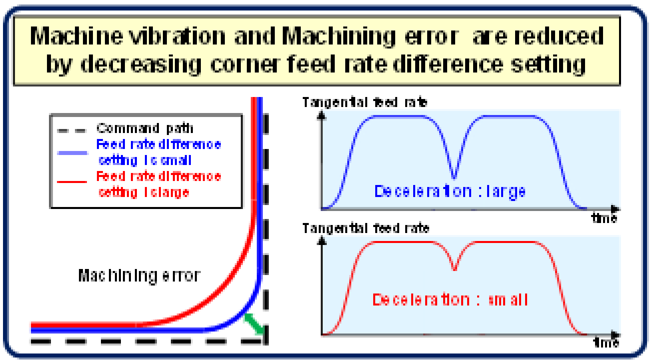
Corner feed rate difference

In the corner, tangential feed-rate curve is smooth, but feed-rate change of each axis is large, so it might cause machine vibration. In this case, machine vibration can be reduced by deceleration with corner feed-rate difference (No.1783) in the corner.
Deceleration feed-rate is decided so that there is no difference of setting value of corner feed-rate difference for each axis or more.
Feed-rate difference under the value of setting corner feed-rate difference in the corner can be smooth by Acc./Dec. after interpolation. And machining error is reduced by reducing feed-rate difference in the corner.



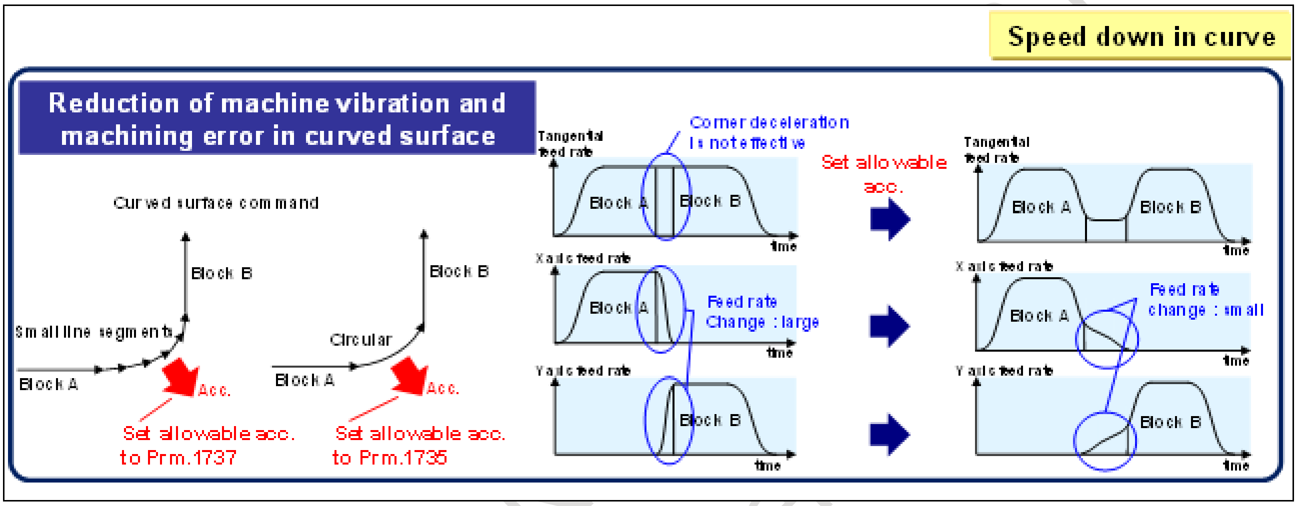
Feed-rate in the corner is decided by parameter setting value of corner feed-rate difference and corner angle.
In case of the same parameter setting value of corner feed-rate difference, if corner angle is smaller, feed-rate difference of each axis is made below parameter setting value by more deceleration in the corner.
In case of the same corner angle, if parameter setting value of corner feed-rate difference is smaller, machine vibration and machining error are reduced by more deceleration in the corner.



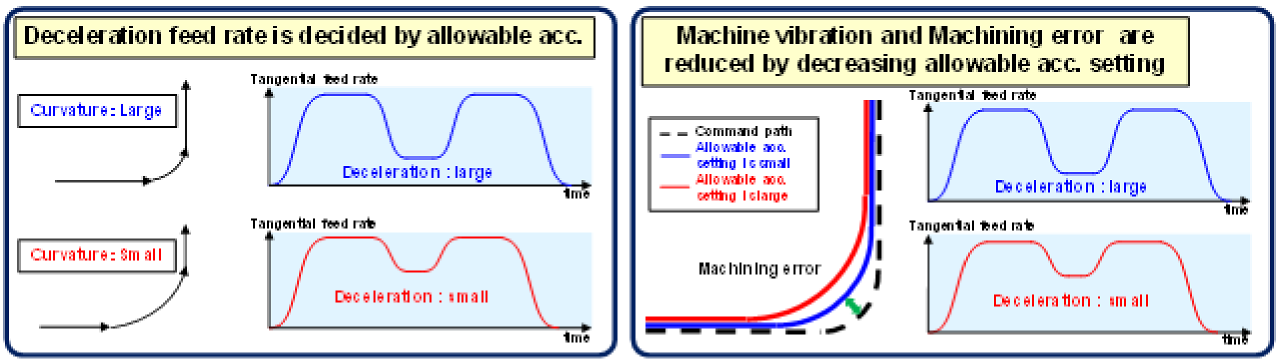


Allowable acceleration rate for curved surface

In the curved surface, tangential feed-rate curve is smooth, but feed-rate change (=acceleration rate) of each axis is large, so it might cause machine vibration. Corner deceleration is not effective in the curved surface. Even if the curved surface is composed of small line segment blocks, corner deceleration is not effective because feed-rate difference between blocks is small.
In this case, machine vibration can be reduced by limiting acceleration rate with allowable acceleration rate (No.1735, No.1737) in the curved surface.
Moreover, as corner deceleration, machining error caused by Acc./Dec. after interpolation is reduced by deceleration in the curved surface.
Different allowable acceleration rate value can be set for small line segment blocks (No.1737) and circular interpolation (No.1735).



Feed-rate in the curved surface is decided by parameter setting value of allowable acceleration and curved rate.
In case of the same parameter setting value of allowable acceleration, if curved rate is larger, acceleration rate of each axis is made below parameter setting value by more deceleration in the curved surface.
In case of the same curved rate, if parameter setting value of allowable acceleration is smaller, machine vibration and machining error are reduced by more deceleration in the curved surface.



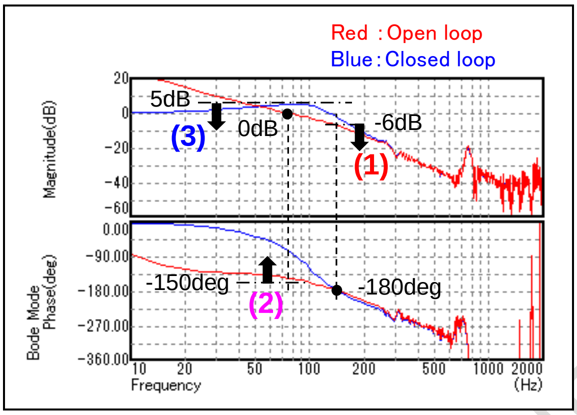
Supplement B. Servo tuning for higher response

Tuning for higher servo stiffness and for improvement of machining accuracy.

Tuning of velocity Loop gain and filter

Measure frequency response by SERVO GUIDE and increase velocity loop gain with keeping the following three conditions.

(1) Open loop gain below -6dB
where phase = -180 deg
(2) Open loop phase above -150 deg
where gain = 0dB
(3) Closed loop gain below +5dB



When vibration occurs, refer to the following table.

Vibration frequency	Method of tuning	Standard parameter range of filter
Lower 200Hz	Decrease velocity loop gain	—
200 to 400Hz	Apply resonance elimination filter	Attenuation bandwidth : 60 to 100Hz Damping : 0 to 50%
400 to 1000Hz	Apply resonance elimination filter	Attenuation bandwidth : 100 to 200Hz Damping : 0 to 10%
Upper 1000Hz	Apply resonance elimination filter	Attenuation bandwidth : 300 to 600Hz Damping : 0 to 10%

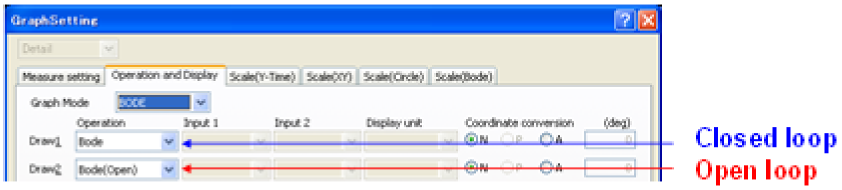
Tuning of velocity feed-forward

Setting of feed-forward coefficient calculated by following formula

Parameter number	Inertia information	No inertia information
No.2069 No.2145	$100 \times \dfrac{\text{Rotor inertia of motor} + \text{Load inertia}}{\text{Rotor inertia of motor}}$	100

[Supplement 1] Display of open loop characteristic

Push "F5", display "Operation and display" and select Bode(Open) in Draw2's operation.



[Supplement 2] Measurement of vibration over 1kHz

Measure TCMD with 125μs sampling cycle.

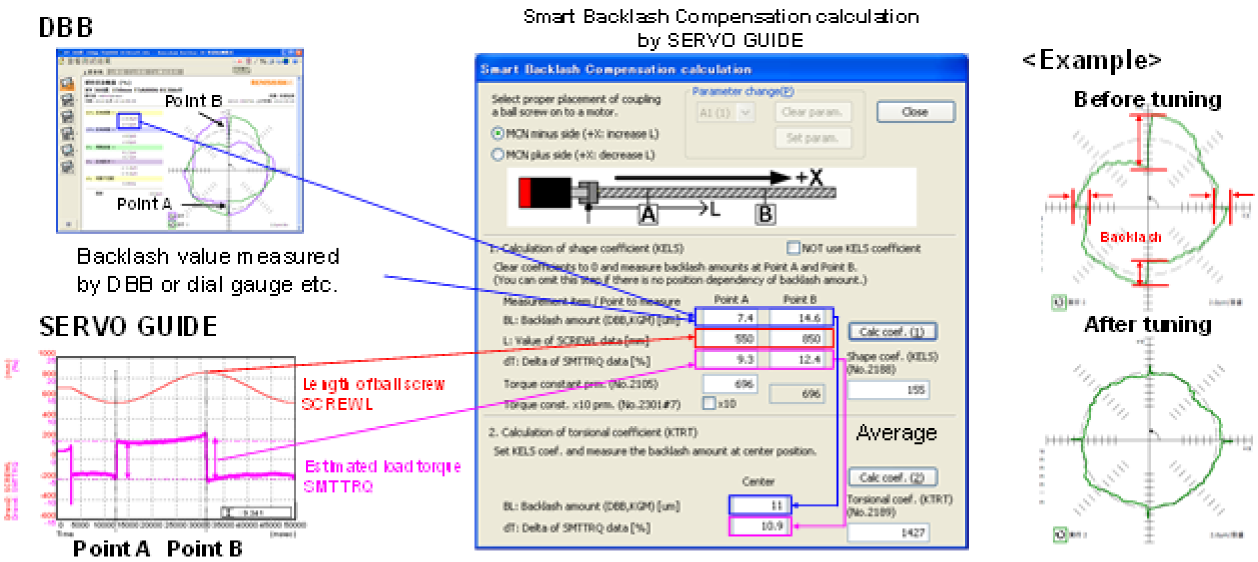
Tune range on YT mode.
(Zoom up or down on time range by cursor key up or down
and decrease wave number below 10 waves per 1 division.)

Do Fourier analysis by "Ctrl + F" , set wave information check box(w-info),
and measure peak frequency.

Tuning of Smart Backlash Comp.

Tuning of Smart Backlash Compensation with DBB and SERVO GUIDE

1. Measure backlash by DBB and measure length of ball screw and estimated load torque by SERVO GUIDE
2. Set measured backlash (Backlash is able to be measured by dial gauge etc. instead of DBB.)
3. Set length of ball screw and change value of estimated load torque when axis reversing



E AUTOMATIC SETTING / INITIAL SETTING PARAMETER

E.1 AUTOMATIC SETTING OF AUTOMATIC SETTING / INITIAL SETTING PARAMETER

Some parameters are automatically set to recommended values in Series 0*i*-F Plus, so that you can use the CNC by recommended setting.
These parameters set up automatically are classified into following (1) or (2).

(1) Automatic setting parameter
(2) Initial setting parameter

The specifications of (1) and (2) are as follows.

(1) Automatic setting parameter

The parameter value is compared with the initial value at power-on. If the value is other than the initial value, the parameter is reset to the initial value.

The parameter value cannot be changed by the parameter screen, PMC window function (WINDW: SUB52), FOCAS2, C Language Executor, Macro Executor, and programmable parameter input (G10), because it is basically unnecessary for the machine tool builder to change the parameter value.

(2) Initial setting parameter

The parameter is set to the recommendation setting value at power-on only when memory all clear has been executed.
Although use with recommended value is recommended, the machine tool builder can change the parameter value.

The parameter of (1) and (2) is set to recommended value at the time of shipment of CNC.

Prohibition of setting to automatic setting parameter

If automatic setting parameter is tried to set to other than recommended value, the following operations are performed.

- Setting parameter from parameter screen
  The warning message "WRITE PROTECTED" is displayed, and the parameter value is not changed.

- PMC window function (WINDW: SUB52)
  Completion code 7 (write-protected) is returned, and the parameter value is not changed.

- FOCAS2, C Language Executor (cnc_wrparam, cnc_wrparas)
  In the case of setting by cnc_wrparam, return value is set to EW_PROT (Writing protection) and the parameter value is not changed.
  In the case of setting by cnc_wrparas, return value is set to EW_PROT (Writing protection), but the values of the parameters before prohibited one is changed.

- Macro Executor (G314)
  Completion code 7 (The data of the parameter corresponding to the specified parameter number is protected.) is returned, and the parameter value is not changed.

- Programmable parameter input (G10)
  The alarm PS1302, "ILLEGAL DATA NUMBER" occurs, and the parameter value is not changed.

Operation when the parameter file is inputted

When the parameter file is inputted, automatic setting parameter is also rewritten according to the contents of the parameter file.
Whether automatic setting parameters are automatically set up again at the time of next power-on.

Automatic servo parameter initialization

There are standard parameters for the servo motor in automatic setting parameters. (feed- forward setting, HRV3 control setting)
When automatic servo parameter initialization (after bit 1 of parameter No.2000 is 0, power is turned off and turned on.) is executed, feed-forward settings(No.2005#1, No.2069, No.2092) are set automatically.

Automatic spindle parameter initialization

There are spindle parameters in automatic setting parameters. Some parameters are that the initial setting value of automatic spindle parameter initialization is different from the recommended value of this function.
Priority is given to the recommended value of this function in Series 0*i*-F Plus.

When automatic spindle parameter initialization (MAIN) (bit 7 of parameter No.4019 is 1) is executed, the following parameters are set to the recommended values of this function.
No.4542#4, No.4627

When automatic spindle parameter initialization (SUB) (bit 7 of parameter No.4195 is 1) is executed, the following parameters are set to the recommended values of this function.
No.4344

E.2 AUTOMATIC SETTING PARAMETER

The list of automatic setting parameter is shown below.

Parameter No.	Meaning of parameters	Automatic setting value
No.301#1	When the input from the connected device to NC is aborted via Ethernet with CNC screen display, 0: Alarm has been occurred. 1: Alarm is not occurred.	1
No.909#0	On the Data server host file list screen, high speed file search is: 0: Invalid. 1: Valid.	1
No.1611#1	When a threading command is specified in AI contour control mode: 0: An alarm is issued. 1: AI contour control mode is temporarily canceled and the command is executed.	1
No.2013#0	HRV3 current control is: 0: Not used. 1: Used.	1 *Only for the M Series
No.2283#0	The high-speed HRV current control mode (servo HRV3 control) is: 0: Set only when both G5.4Q1 and G01 are specified. 1: Set when G01 is specified (G5.4Q1 is not monitored).	1 *Only for the M Series
No.3002#2	When the high-speed M/S/T/B interface is used, if a block specifying an M, S, T, or B code does not contain a move command or dwell command, the distribution end signal and the strobe signal for the function are: 0: Output conventionally (the output of the distribution end signal is delayed). 1: Output at the same time.	1
No.3458#1	In retrace function, when reverse execution is done during auxiliary function execution and then forward reexecution is done, the output of auxiliary function is: 0: Once. 1: Twice.	1 *Only for the M Series
No.3470#0	In helical interpolation, when the circular interpolation axes don't move and other specified axes don't move: 0: Alarm PS0021,"ILLEGAL PLANE SELECT" is issued. 1: Alarm is not issued.	1
No.3729#2	When the Cs contour control mode is turned off, an in-position check is: 0: Performed. 1: Not performed.	1
No.3785#2	At the start of cutting feed, : 0: Always wait until the time elapsed prior to checking the spindle speed arrival signal of parameter No. 3740. 1: If the time longer than the parameter No. 3740 has elapsed since the spindle speed was changed, it does not wait for the passage of the parameter No.3740.	1
No.3785#7	The speed-up of changing Cs contour control mode is 0: Disabled. 1: Enabled.	1
No.5107#0	For G71/G72 (G-code system A), movement to the last turning start position is performed by: 0: Cutting feed. 1: Rapid traverse.	1 *Only for the T Series

Parameter No.	Meaning of parameters	Automatic setting value
No.5107#1	The G71/G72 (G-code system A) TYPE1 commands execute the movement toward the current turning start position in: 0: Two cycles. 1: One cycle.	1 *Only for the T Series
No.5107#2	In G70-G73 (G-code system A), the cycle operation is executed by: 0: Conventional method. 1: Improved method.	1 *Only for the T Series
No.5107#5	If Multiple repetitive canned cycle (G70 to G76) is specified in Coordinate system rotation, Programmable mirror image or Scaling mode: 0: Alarm is not issued. 1: Alarm (PS0327) "MODAL THAT MULTIPLE REPETIVE CYCLES CANNOT BE DONE" is issued.	1 *Only for the T Series
No.5108#3	In the multiple repetitive cycle G71/G72 (G-code system A) of type II, the cutting is executed: 0: By conventional path. (The same cutting path might be executed.) 1: Not to repeat the same cutting path.	1 *Only for the T Series
No.5108#5	In Multiple repetitive cycle G71/G72 (G-code system A), in-position check between cutting blocks is: 0: Executed. 1: Not executed.	1 *Only for the T Series
No.5109#0	When an axis, which is not included in the specified plane, is commanded in the multiple repetitive cycle (G70 to G76): 0: An alarm does not occur. 1: An alarm PS0021,"ILLEGAL PLANE SELECT" occurs.	1 *Only for the T Series
No.5124#0	In a TYPE-1 of multiple repetitive canned cycle (G71, G72), escape after rough cutting is performed by: 0: Cutting feed. 1: Rapid traverse.(The tool path follows the setting in bit 1 (LRP) of parameter No. 1401.)	1 *Only for the T Series
No.5125#6	In type I of G71, G72 of multiple repetitive canned cycle, the axes : 0: Don't move directly from retraction end point to the last turning start point. 1: Move directly from retraction end point to the last turning start point.	1 *Only for the T Series
No.5203#2	In rigid tapping, feed forward is: 0: Disabled. 1: Enabled.	1
No.5208#0	High-speed rigid tapping is: 0: Invalid. 1: Valid.	1
No.5208#1	Speed-up of the motion at the hole bottom in rigid tapping is: 0: Disabled. 1: Enabled.	1
No.5208#5	Speed-up of cycle operation in rigid tapping is: 0: Disabled. 1: Enabled.	1
No.5209#6	If Constant surface speed control is commanded in Rigid tapping mode: 0: Alarm is not issued. 1: Alarm (PS0200) "ILLEGAL S CODE COMMAND" is issued.	1
No.5209#7	On the peck rigid tapping, if the depth of cut (Q) is smaller than the cutting start distance (d): 0: Alarm is not issued. 1: Alarm (PS5560) "ILLEGAL DEPTH OF CUT" is issued.	1

Parameter No.	Meaning of parameters	Automatic setting value
No.5402#6	In case that Positioning has not been done after Coordinate system rotation, if Circular interpolation with R is specified: 0: a center angle might change by over 180 degrees. 1: a center angle does not change by over 180 degrees.	1
No.6000#2	A custom macro is executed: 0: At a normal speed. 1: At a high-speed.	1
No.6021#1	For rotation axes for which rollover function is enabled, the reading of block end point position by system variables #5001 to #5020 or #100001 to #100050 is: 0: Not available. 1: Available.	1
No.8019#1	After the move command of PMC axis control is interrupted by reset signal ECLRg, If different acceleration/deceleration type command without waiting for a deceleration stop generates: 0: Not alarm. 1: Alarm DS1451,"IMPROPER PMC AXIS COMMAND".	1
No.8019#2	If the controlled axis selection signals EAX1 to EAX8 are "0" or PMC controlled-axis selection variable (#8700) is 0, the PMC axis control command generates: 0: Not alarm. 1: Alarm DS1451 "IMPROPER PMC AXIS COMMAND".	1
No.8106#2	By reset in the path other than the path 1, the custom macro variables common to tool path set to common variables #100-#499 cleared by power-off are: 0: Not cleared to <null>. 1: Cleared to <null>. (The setting bit 6 (CCV) of parameter No. 6001 is depended on.)	1
No.8209#4	When the reference position return of angular axis is performed in the slanted coordinate system the reference position return of perpendicular axis is: 0: Not performed in the slanted coordinate system. 1: Performed in the slanted coordinate system.	1
No.8655#3	When the crt_cncscrn function is executed from the main task of C Language Executor, the main task is: 0: Not stopped at the end of the function (is stopped after processing is performed for a very short time). 1: Stopped at the end of the function.	1
No.9032#6	Speed-up read/write function of P-CODE variables(#10000～#19999) and extended P-CODE variables(#20000～#89999) is: 0: Enabled. 1: Disabled.	0
No.10345#1	When the forbidden area of the stored stroke check 2, 3 (Parameter No.1322, 1323) is set or it is changed by G22 command, the setting value for axes with diameter specification is: 0: Half of command value 1: Command value	1
No.10350#1	The servo loop gain for each axis (parameter No. 1825) and the In-position width for each axis (parameter No. 1826) are: 0: Write-disabled during axis moving. 1: Write-enabled if the corresponding axis is stopped.	1

Parameter No.	Meaning of parameters	Automatic setting value
No.10597#0	On the Dual Check Safety Diagnosis screen, the following information is: 0: Not displayed. 1: Displayed. - Safety speed limit (unit: mm/min, inch/min, deg/min) - Safety speed zero monitoring request signal. - Safe feed limit/safe machine position selection signal - Safe feed limit/safe machine position selection signal in case of an alarm - Programmable Safety I/O signal	1
No.11002#1	Speed-up of Spindle indexing is: 0: Disabled. 1: Enabled.	1
No.11303#4	During MDI program execution, blocks that call an execution macro are: 0: Not displayed. 1: Displayed.	1
No.11353#5	While user program which is called by Execution macro function is executing, the data returned by FOCAS2 function as the data under execution is: 0: Data of user program. 1: Data of caller program.	1
No.11354#1	While data transmission is awaited using the DPRNT/BPRNT of the custom macro or macro executor, screen switching is: 0: Not possible. 1: Possible.	1
No.11354#2	When the storage capacity for history data is exceeded due to non-alarm history, alarm history will be: 0: Erased. 1: Erased, except the most recent 50 items of history data.	1
No.11354#3	When the storage capacity for history data is exceeded due to data other than external operator message history, external operator message history will be: 0: Erased. 1: Retained.	1
No.11374#0	If EOB code is included in comment block when program is read: 0: Alarms are not generated. 1: The alarm PS0518,"CODE OF CONTROL-IN DOES NOT EXIST" is generated.	1
No.11400#3	Select the behavior of following system variable related to tool length offset by reset operation. #5081-#5100 in case of the parameters LVC(No.5006#3)=1, LWT(No.5002#2)=0. #5121-#5140 in case of the parameters LVC(No.5006#3)=1, LGT(No.5002#4)=1. 0: Not clear the system variable. 1: Clear the system variable.	1 *Only for the T Series
No.11500#1	Cutting feed-forward flag is 0: output immediately when it comes in-position state. 1: output after waiting for 1 interpolation period after becoming in-position state.	0
No.11501#5	The timing of starting detection of a one-rotation signal in threading is: 0: conventional. 1: high-speed execution.	1

Parameter No.	Meaning of parameters	Automatic setting value
No.11501#6	Speed-up of G28, G30 and G53 is: 0: Disabled. 1: Enabled.	1
No.11502#3	If the cycle start of MEM/RMT mode is commanded without reset while executing subprogram/macro call of the MDI mode: 0: The alarm is not generated. 1: The alarm PS0525,"subprogram/macro calling." is generated.	1
No.11503#2	The precision improvement of the actual machine position by speed up of the timing that the servo position deviation is reflected is: 0: Invalid. 1: Valid.	1
No.11503#4	When the axis controlled with a servo motor enters in-position check: 0: The next block is begun at the next interpolation cycle that entered in-position. 1: The next block is begun at the interpolation cycle when entering in-position.	1
No.11504#5	Operating time reduction of command of preventing buffering is: 0: Disabled. 1: Enabled.	1
No.11507#3	Speed-up of processing related to RS-232C is: 0: Disabled. 1: Enabled.	1
No.11538#6	In automatic operation, the speed-up between blocks is: 0: Invalid. 1: Valid.	1
No.11565#1	In Flexible path axis assignment, tool geometry offset value and tool wear offset value and tool offset value are: 0: not canceled and alarm PS0514 occurs when flexible path axis assignment command is issued. 1: canceled only for the commanded axis when flexible path axis command is issued. In this case, the axis does not move regardless of the tool offset methods.	1 *Only for the T Series
No.11602#2	In the continuous block of the high-speed G53, in the case of interpolation type rapid traverse or time fixed type, the rapid overlap is: 0: Invalid. 1: Valid.	1
No.11602#3	When the rapid traverse is linear interpolation type and time constant acceleration/decerelation type, the rapid overlap is: 0: Invalid. 1: Valid.	1
No.11620#0	In the program in which the cutting feed (G01) in AI contour control mode is continuous, when there is a block with the same position command in between,: 0: It decelerates and stops. 1: Ignore the block of the same position command and do not decelerate and stop, overlapping with the next block.	1
No.11651#1	When GOTO statement using stored sequence numbers/LGOTO statement using stored label is enabled and memory operation of a program on Data Server/Memory card is executed, sequence numbers/LBL[label] in the program are: 0: Stored. 1: Not stored.	1

Parameter No.	Meaning of parameters	Automatic setting value
No.11850#0	If in PMC axis control, a rapid traverse rate is specified with the axis control block data signal, with bit 0 (RPD) of parameter No. 8002 being set to 1, the rapid traverse rate is: 0: Always treated as being in millimeters. 1: Dependent on the setting of bit 0 (INM) of parameter No.1001.	1
No.13205#5	In Tool management function, when following G codes and M6 are commanded in the same block, axes move to the end position of the block: - G53 Machine coordinate system setting (except for high speed G53) - G28 Automatic return to reference position - G29 Movement from reference position - G30 2nd, 3rd and 4th reference position return 0: After FIN signal returned. 1: Before FIN signal returned.	1
No.13208#6	In case of tool management function, if T code is commanded when either of following PMC window is executing, edge number change and tool search by T code command are: - PMC window for tool exchange (Function code 329) - PMC window for tool moving (Function code 329) - PMC window for shifting tool management data (Function code 367) 0: Not kept waiting. 1: Kept waiting until PMC window is finished.	1
No.13421#2	If G27/G28/G29/G30/G53 is specified during flexible synchronization control, alarm PS0010,"IMPROPER G-CODE" is: 0: Issued. 1: Is not issued. Commands to the master axis are possible.	1
No.13421#4	For a synchronization group for which a PMC axis is a master axis, when the controlled axis selection signal EAXn<G0136> for PMC axis control is set to "1" after starting flexible synchronization control mode, and the master axis is specified by PMC axis control, or for a synchronization group for which spindle control with servo motor is a master axis, when SV speed control mode of the master axis is turned on after starting flexible synchronization control mode: 0: Alarm is not generated. 1: Alarm PS5381 "INVALID COMMAND IN FSC MODE" is generated.	1
No.13450#4	When the cutting is executed without specifying a feedrate (F) after the modal G code of group 05 was changed by G93(inverse time feed) / G94(feed per minute) / G95(feed per revolution) command: 0: The feedrate (F) is inherited as a modal. 1: Alarm PS0011,"FEED ZERO (COMMAND)" is issued.	1
No.13805#1	When the setting of that follow-up is executed by follow-up signal (*FLWU) in servo-off state is effective (the value of the parameter FUPx(No.1819#0) is 0), the following limit is applied to the safety position error monitoring function during servo off state and follow-up state including emergency stop and servo alarm etc, or pole position detection . 0: The position deviation limit in moving state (The parameter No.1838, 1841) 1: The position deviation limit in servo-off state (The parameter No.1840)	1
No.13806#0	The brake signal(*BRKx) of Dual Check Safety in servo off state 0: is not to 0. 1: is to 0.	1

Parameter No.	Meaning of parameters	Automatic setting value
No.18050#7	If an axis move command is executed with PMC axis control during automatic operation, and the NC block under execution is stopped by a feed hold when the axis moving due to PMC axis control is completed, the amount of movement due to PMC axis control in that block is: 0: Not reflected in the NC coordinate system. 1: Reflected in the NC coordinate system.	1
No.24308#4	While the execution macro is being executed, making the data for displaying program being executed is: 0: processing 1: not processing	1

E.3 INITIAL SETTING PARAMETER

The list of initial setting parameter is shown below.

Parameter No.	Meaning of parameters	Initial setting value
No.101#0	The number of stop bits 0: 1 1: 2	1
No.101#7	Feed before and after the data output 0: Output 1: Not output	1
No.103	Baud rate (when I/O CHANNEL is set to 0)	10
No.1602#3	Acceleration/deceleration after interpolation for cutting feed in a mode of look-ahead acceleration/deceleration before interpolation such as the AI contour control mode: 0: Exponential acceleration/deceleration or linear acceleration/ deceleration is used. (The setting of bit 6 (LS2) of parameter No. 1602 is followed.) 1: Bell-shaped acceleration/deceleration is used. (The bell-shaped acceleration/deceleration after cutting feed interpolation option is required.)	1
No.1602#6	Acceleration/deceleration after interpolation for cutting feed in a mode of acceleration/deceleration before interpolation such as the AI contour control mode: 0: Exponential acceleration/deceleration is used. 1: Linear acceleration/deceleration is used.	1
No.1604#0	When automatic operation is started, the state equivalent to the specification of G5.1Q1 for AI contour control is: 0: Not set 1: Set	1 *Only for the M Series
No.1610#0	Acceleration/deceleration in cutting feed or dry run during cutting feed 0: Exponential acceleration/deceleration is applied. 1: Linear acceleration/deceleration after interpolation is applied.	1
No.1610#1	Acceleration/deceleration in cutting feed or dry run during cutting feed 0: Exponential acceleration/deceleration or linear acceleration/ deceleration is applied. (depending on the setting in bit 0 (CTLx) of parameter No. 1610) 1: Bell-shaped acceleration/deceleration is applied.	1

Parameter No.	Meaning of parameters	Initial setting value
No.1610#4	Acceleration/deceleration in jog feed 0: Exponential acceleration/deceleration is applied. 1: The same acceleration/deceleration as for cutting feedrate is applied. (Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)	1
No.1660	Maximum allowable acceleration rate in acceleration/deceleration before interpolation for each axis	2000.0 *Only for the M Series
No.1800#3	Feed-forward control in rapid traverse is: 0: Disabled. 1: Enabled.	1
No.1825	Position loop gain	3000
No.2005#1	Feed-forward function is: 0: Disabled. 1: Enabled.	1
No.2013#0	HRV3 current control is: 0: Not used. 1: Used.	1 *Only for the T Series
No.2017#7	Velocity loop high cycle management function is: 0: Disabled. 1: Enabled.	1
No.2069	Velocity feed-forward coefficient for rapid traverse	100
No.2092	Advanced preview feed-forward coefficient for rapid traverse	9800
No.2144	Position advanced preview feed-forward coefficient for cutting	10000
No.2145	Velocity feed-forward coefficient for cutting	100
No.2202#1	The cutting/rapid velocity loop gain switching function is: 0: Disabled. 1: Enabled.	1
No.2214#4	The cutting/rapid feed-forward switching function is: 0: Disabled. 1: Enabled.	1
No.2283#0	The high-speed HRV current control mode (servo HRV3 control) is: 0: Set only when both G5.4Q1 and G01 are specified. 1: Set when G01 is specified (G5.4Q1 is not monitored).	1 *Only for the T Series
No.2334	Current loop gain magnification in high-speed HRV current control mode	150
No.2335	Velocity loop gain magnification in high-speed HRV current control mode	200
No.2415#1	The default value of the feed-forward timing adjustment parameter is: 0: 0. 1: Compatible with that of 16 *i* Series.	1
No.3205#5	When the background editing is started without inputting the program name in the key: 0: Program to be edited are initialized (to the state where no selection is made). 1: the editing of the previously edited programs are continued. (Continued editing is possible only when last edited time is not changed (to allow continued editing).)	1
No.3403#6	When the same address two or more times are specified in one block: 0: The address specified last is valid. 1: It is treated as a program error and the alarm PS5074, "ADDRESS DUPLICATION ERROR" is issued.	1
No.4344	Advanced feed-forward coefficient	9800

Parameter No.	Meaning of parameters	Initial setting value
No.4542#4	The cutting feed/rapid traverse feed-forward function is: 0: Disabled. 1: Enabled.	1
No.4627	Advanced preview feed-forward coefficient for cutting feed	10000
No.5203#5	As acceleration/deceleration for rigid tapping cutting feed: 0: Linear acceleration/deceleration is used. 1: Bell-shaped acceleration/deceleration is used.	1
No.5503#0	In the index table indexing function, when the parameter ABS(No.5500#2) is set to 1, internal coordinate system is: 0: Not rounded by one rotation. 1: Rounded by one rotation.	1 *Only for the M Series
No.7003#0	When the manual absolute is on and manual operation is executed in reset state or automatic operation stop state, the movement amount of the manual operation is: 0: Reflected to the movement amount of the first absolute command. 1: Reflected to the coordinate system at the cycle start.	1
No.8306#0	In synchronization establishment: 0: A machine coordinate difference between the master axis and slave axis is output at a time as command pulses (axis movements are performed without acceleration/deceleration). 1: Axis movements are executed with the feedrate of manual rapid traverse and the acceleration/deceleration after interpolation in rapid traverse.	1
No.11304#0	When the path select signal is changed, the screen of the multi path simultaneous display group: 0: Is not switched. 1: Is switched to the display group including the selected path.	1
No.11308#1	When the file of specified name already exists on memory card: 0: It is not overwritten. Alarm SR1973, “FILE ALREADY EXIST” is generated. 1: It is overwritten.	1
No.11364#3	If a program or a folder exists in the target folder when the deletion operation is done specifying the folder: 0: The folder is not deleted. 1: The folder and programs/folders in the target folder are deleted.	1
No.11391#7	In direct input of offset value measured B for 2 spindle lathe, when the setting that a workpiece coordinate system shift amount is set in the workpiece coordinate system memory is an automatic selection is made (parameter 2AT(No.5051#1), and workpiece coordinate system shift value write mode select signal WOQSM <Gn039.6> is 1, the cursor movement by the MDI operation is: 0: Not prohibited. 1: Prohibited.	1 *Only for the T Series
No.11501#3	At a start of the automatic operation, the time from a falling edge of the cycle start signal ST to a start of the automatic operation is: 0: conventional. 1: reduced.	1
No.11933#0	Set a communication method for the I/O Link channel 1: 0: The I/O Link is used. 1: The I/O Link i is used.	1
No.11933#1	Set a communication method for the I/O Link channel 2: 0: The I/O Link is used. 1: The I/O Link i is used.	1

Parameter No.	Meaning of parameters	Initial setting value
No.14000#1	If an inch-metric switch command is executed at a position other than 0 of the machine coordinate, 0: Alarm is not issued. 1: Alarm PS5362,"CONVERT INCH/MM AT REF-POS" is issued.	1
No.19503#0	When a feedrate is determined based on acceleration in AI contour control, smooth feedrate control is: 0: Not used. 1: Used.	1 *Only for the M Series
No.19517#0	When smooth speed control is effective, by block length of the linear interpolation, smooth speed control and speed control with change of acceleration on each axis; 0: Are not invalidated. 1: Are invalidated.	1 *Only for the M Series
No.19517#1	By block length of the linear interpolation, speed control with acceleration on each axis and speed control with change of acceleration on each axis: 0: Are not invalidated. 1: Are invalidated.	1 *Only for the M Series
No.19800#0	Command path correction function is: 0: Disabled. 1: Enabled.	1 *Only for the M Series
No.24306#2	On the servo tuning screen, path is changed to other path and returned previous path, axis page and cursor position is: 0: Not restored. 1: Restored.	1
No.24311#7	When virtual MDI key function is used, the current position on program screen of TEACH IN mode is: 0: Not displayed. 1: Displayed.	1

F 0*i*-F PLUS START-UP PROCEDURE

In 0*i*-F Plus, Fast cycle-time setting and Fine surface setting are standard. In Fast cycle-time setting, recommended values can be set for parameters related to reducing the cycle time all at once. In Fine surface setting, recommended values can be set for parameters related to high-quality processing all at once. In order to improve the performance of CNC, set the recommended values all at once by these functions.

> **NOTE**
> This function is for the machine tool builder. It is used only when parameter setting at machine start-up.

Blinking of "ADJUST"

In 0*i*-F Plus, in order to urge adjustment by Fast cycle-time setting and Fine surface setting, "ADJUST" blinks in the status section when any function is incompletely adjusted. Be sure to set adjustment completion before machine shipment and hide "ADJUST". (For the adjustment completion setting, refer to "Adjustment completion setting" below.)

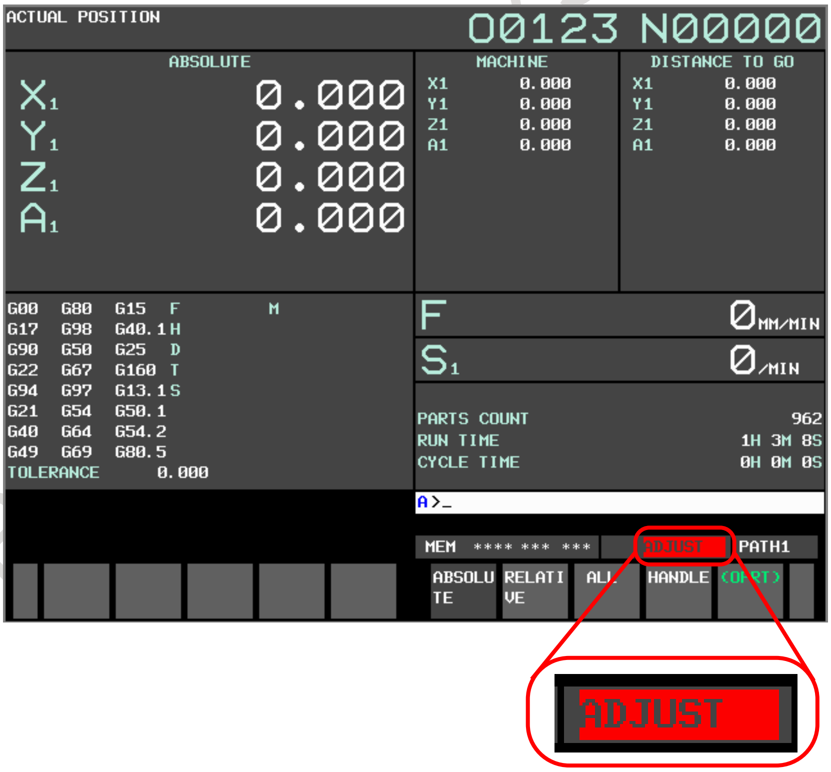


Fig. F (a) Blinking of "ADJUST"

Setting procedure (overview)

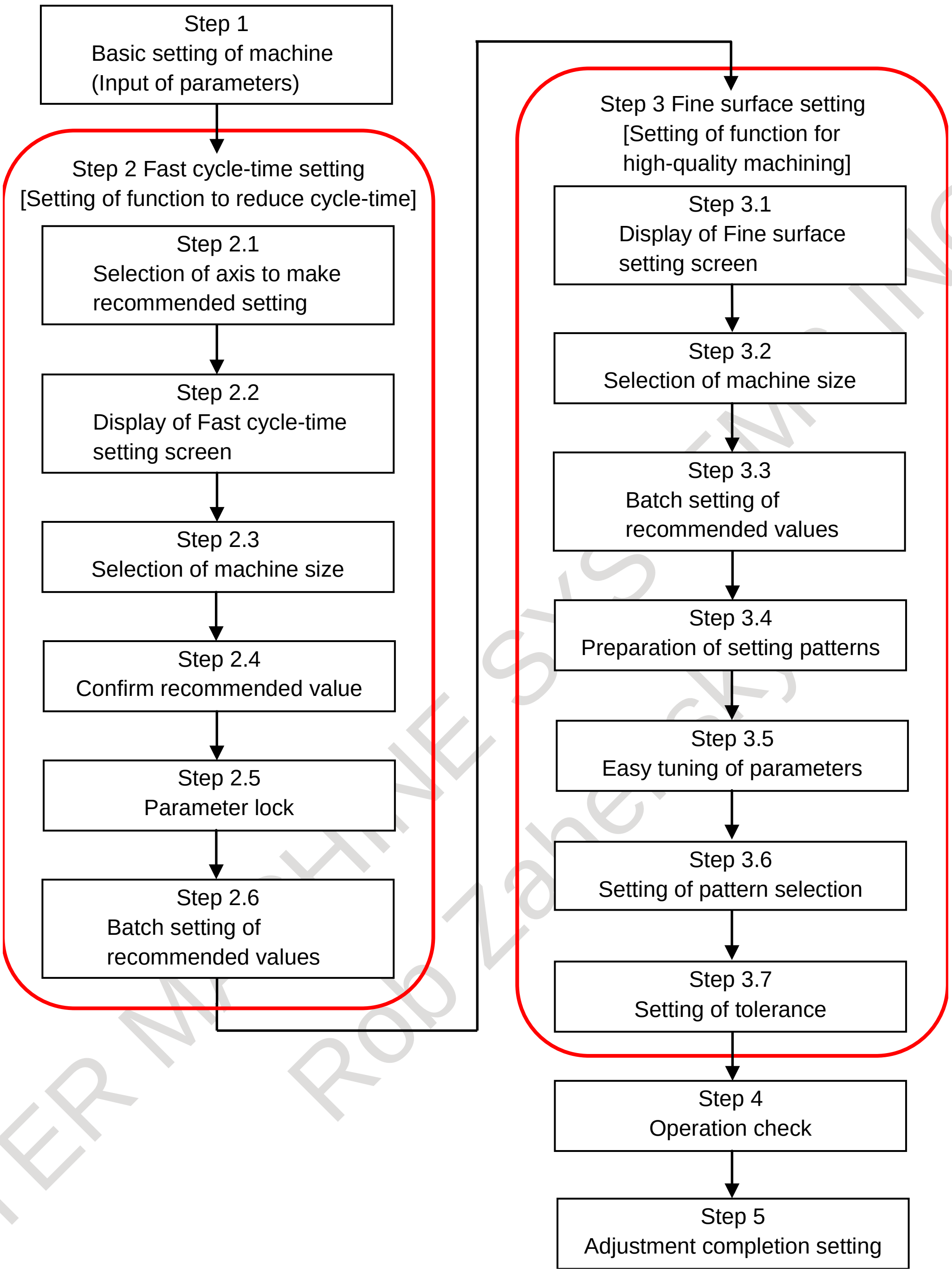


Fig. F (b) Setting procedure (overview)

Below, Step 1 to 5 is explained.

F.1 BASIC SETTING OF MACHINE

Before setting the recommended value by Fast cycle-time setting and Fine surface setting, input the parameters of machine to 0*i*-F Plus and perform basic setting such as number of axes. After inputting the parameters, turn the power OFF/ON.

F.2 FAST CYCLE-TIME SETTING

Overview

Recommended values can be set for parameters related to reducing the cycle time all at once. This makes it easy to set parameters related to reducing the cycle time when the machine is started up.

Explanation

In Fast cycle-time setting, recommended values can be set for parameters related to reducing the cycle time all at once in the following procedure. Setting by Fast cycle-time setting is done on the dedicated Fast cycle-time setting screen (FCS screen).

Step 1
Selection of axis to make recommended setting

↓

Step 2
Display of Fast cycle-time setting screen

↓

Step 3
Selection of machine size

↓

Step 4
Confirm recommended value

↓

Step 5
Parameter lock

↓

Step 6
Batch setting of recommended values

Fig. F.2 (a) Fast cycle-time setting procedure

Below, Step 1 to 6 is explained.

Step 1 : Selection of axis to make recommended setting

The servo axis/spindle that setting recommended values by Fast cycle-time setting is specified. The servo axis is specified by the bit 7 (FAXx) of parameter No.11525. The spindle is specified by the bit 7 (FSPs) of parameter No. 11527. Only the servo axis/spindle with parameter FAXx = 1/FSPs = 1 on the parameter screen becomes the target axis for batch setting. For axes you want to exclude from the batch setting such as the PMC axis, set the parameter FAXx = 0/FSPs = 0.

Step 2 : Display of Fast cycle-time setting screen

The FCS screen can be displayed by pressing the function key SYSTEM when the bit 0 (FCS) of parameter No. 24745 is 0 (adjustment by Fast cycle-time setting is incomplete). On the FCS screen, the selected machine size, item (parameter) name, parameter value, Fanuc standard value, and item (parameter) explanation are displayed. The parameter value is the value on the parameter screen. Fanuc standard value is the standard value set by FANUC (For Fanuc standard value, refer to "Parameter set by Fast cycle-time setting" below). Also, when the parameter FCS is 0, "ADJUST" blinks in the status section.

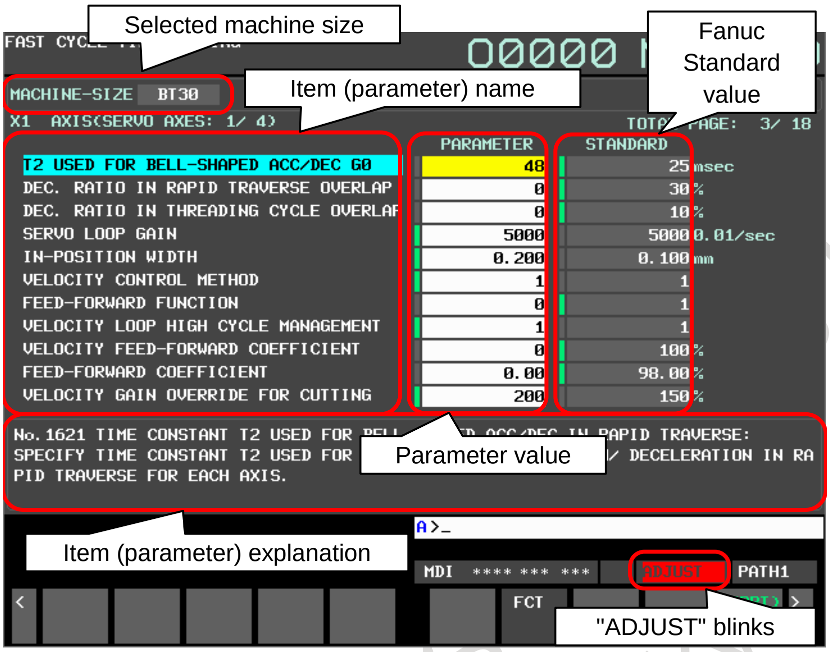


Fig. F.2 (b) Fast cycle-time setting screen (10.4-inch display unit)

There are three major setting items for Fast cycle-time setting.

- “Common”
  System common type, path type parameters related to reducing the cycle time
- “Axis”
  Axis type parameters related to reducing the cycle time
- “Spindle”
  Spindle type parameters related to reducing the cycle time

The screen of each setting item can be displayed by pressing the soft key [COMMON], [CHANGE AXIS], [CHANGE SPINDL].

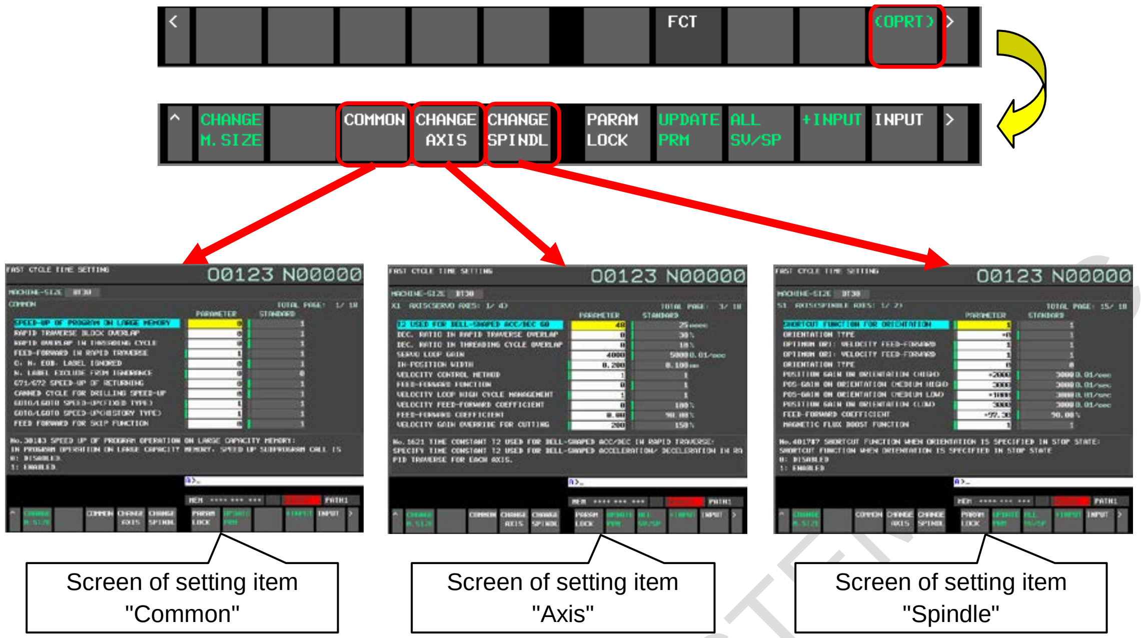


Fig. F.2 (c) Setting item "Common", "Axis", "Spindle"

NOTE

1. The parameter value on the FCS screen differs from the value displayed on the parameter screen and is displayed in general unit.
   Example 1)
   Display contents of parameter No. 1826 when the metric system machine, the detection unit 0.001 (IS-B, CMR = 1)
   Parameter screen 100 [detection unit]
   FCS screen 0.100 [mm]
   Example 2)
   Display contents of parameter No. 2092
   Parameter screen 9800 [0.01%]
   FCS screen 98.00 [%]
2. When the parameter FCS is 0 (when "ADJUST"), the FCS screen is added to the tab stop of the function key SYSTEM. Tab stop means that predetermined screens are displayed in order when the function key is repeatedly pressed.
   Example) The screen displayed when the function key POS is pressed.

Function key	Displayed screen and order
POS	→ Absolute position screen → Relative position screen → Overall position display

Step 3 : Selection of machine size

By pressing the soft key [CHANGE M.SIZE], the machine size can selected from the 3 patterns. Select the appropriate one (or closer one).

0*i*-MF Plus : Spindle taper size [BT30 / BT40 / BT50]

0*i*-TF Plus : Auto lathe and chuck size [AUTO LATHE / 8"CHUCK / 12"CHUCK]

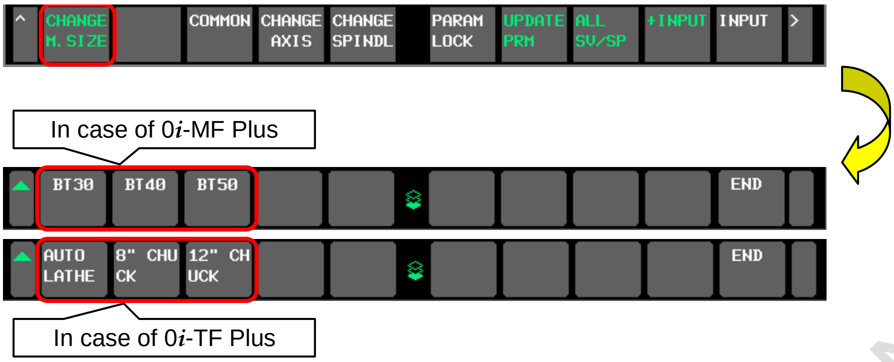


Fig. F.2 (d) Selection of machine size

When changing the machine size, the display of the machine size at the upper left of the screen changes. In addition, the Fanuc standard value also changes to the value set for each machine size.

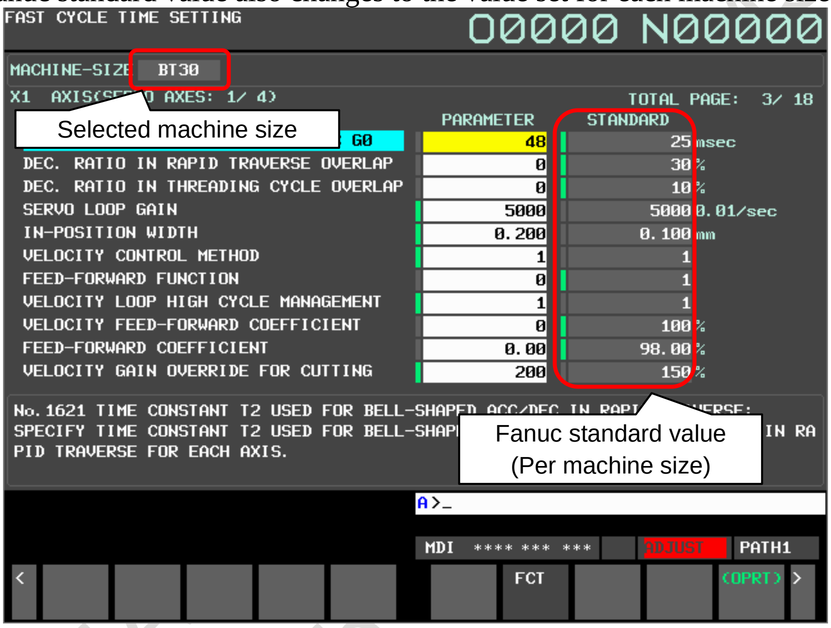


Fig. F.2 (e) Machine size and Fanuc standard value

> **NOTE**
> Machine size is common for fast cycle time setting and fine surface setting. When the machine size is changed on one setting screen, the change is reflected on the other setting screen as well.

Step 4 : Confirm recommended value

When the FCS screen is displayed, CNC compares the current "parameter value" with "FANUC standard value". As a result of the comparison, CNC judges the one with the larger cycle time reducing effect as the recommended value. Also, the green lamp lights up on the side that CNC judged as the recommended value. This allows you to check the current recommended values.

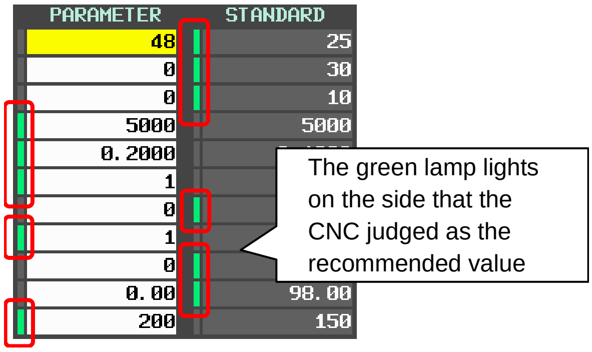


Fig. F.2 (f) Confirm recommended value

Step 5 : Parameter lock

By the soft key [PARAM LOCK], it is possible to become the state (the parameter locked state) that always select the parameter value for each item. By this, the batch setting can be done while keeping the setting unique to the machine tool builder. To set the parameter locked state, move the cursor to the item you want to set and press the soft key [PARAM LOCK]. When set, the red lamp lights up on the parameter side. By this, you can check which item is in the parameter locked state. To cancel the parameter locked state, move the cursor to the item you want to cancel and press the soft key [PARAM UNLOCK]. The soft key [PARAM UNLOCK] is displayed instead of the soft key [PARAM LOCK] when the cursor is on the item in the parameter locked state.

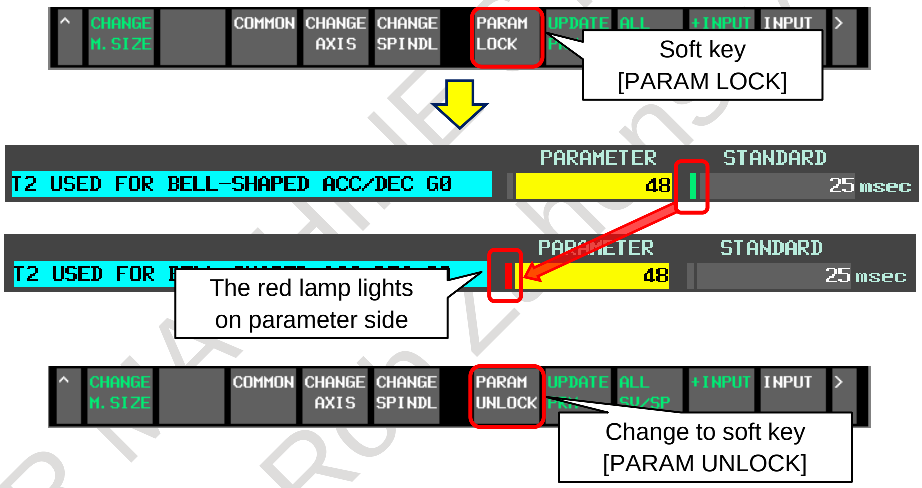


Fig. F.2 (g) Parameter lock

Step 6 : Batch setting of recommended values

When the soft key [UPDATE PRM] is pressed, the parameter values of all items (items of "common", "axis", "spindle" of all paths) are recommended values (values in which the lamp is lit) . (The standard value on which the lamp is lit is set to the parameter value all at once.)

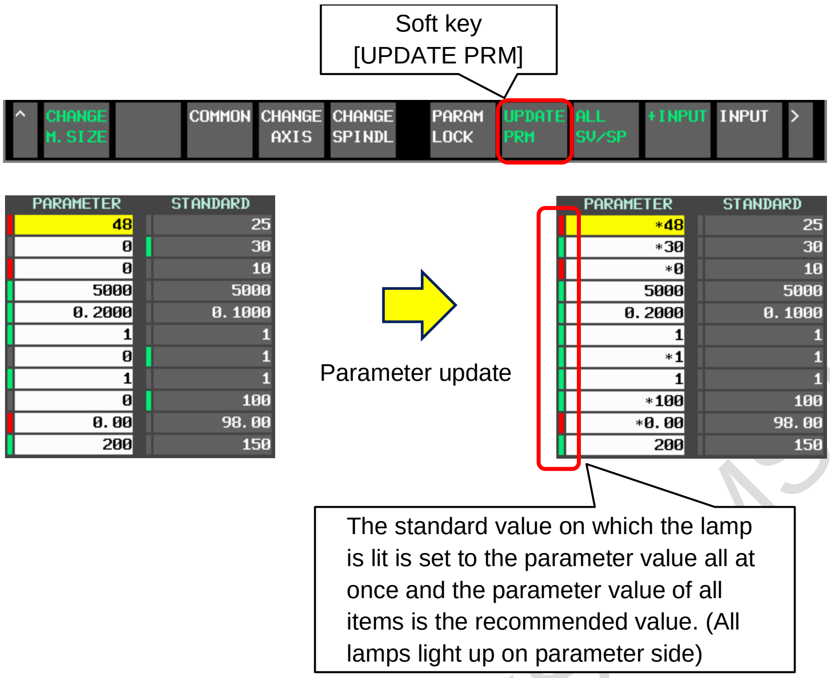
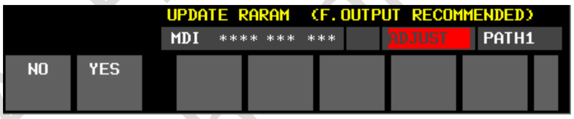


Fig. F.2 (h) Batch setting of recommended values

NOTE

1. The lamp only shows which parameter value or Fanuc standard value is selected as the recommended value. The operation of the machine operates according to the currently set parameter value.
2. The soft key [parameter update] is executed for all items of all paths, not for each path.
3. Items whose values are written to the parameters by the soft key [UPDATE PRM] are only the items whose standard value is the recommended value (items whose lamps are lit on the standard value side). Values are not overwritten for items whose parameter values are already recommended values (items whose lamps are lit on the parameter side).
4. Execute the soft key [UPDATE PRM] in emergency stop or in MDI mode and not in automatic operation. If it is executed in another mode, the warning "WRONG MODE" occurs.
5. The bit 0 (PWE) of parameter No. 8900 is valid for the operation to change the parameter value (direct input, the soft key [UPDATE PRM], etc.). If the write operation is executed with write disable setting (parameter PWE = 0), a warning "WRITE PROTECT " occurs. (8-Level data protection function is invalid. When a write operation is executed with the setting of write enable (parameter PWE = 1), the parameter value is changed regardless of the setting of 8-Level data protection function.)
6. The soft key [UPDATE PRM] changes the parameter value. Therefore, we recommend backup the parameters before change. The parameter input/output (the soft key [F input] [F output]) is possible on the FCS screen as well as the parameter screen. Also, when the soft key [UPDATE PRM] is pressed, the following confirmation guidance is displayed.



Parameters set by Fast cycle-time setting

Parameters set by Fast cycle-time setting are as shown in the table below.

Table F.2 (a) Parameters displayed on the FCS screen "Common"

Item Name / Parameter number	0*i*-MF Plus			0*i*-TF Plus		
	BT30 Standard value	BT40 Standard value	BT50 Standard value	Auto Lathe Standard value	8” Chuck Standard value	12” Chuck Standard value
SPEED-UP OF PROGRAM ON LARGE MEMORY / 301#3	1	1	1	1	1	1
RAPID TRAVERSE BLOCK OVERLAP / 1601#4	1	1	1	1	1	1
RAPID OVERLAP IN THREADING CYCLE / 1611#3	1	1	1	1	1	1

Item Name / Parameter number	0*i*-MF Plus BT30 Standard value	BT40 Standard value	BT50 Standard value	0*i*-TF Plus Auto Lathe Standard value	8” Chuck Standard value	12” Chuck Standard value
FEED-FORWARD IN RAPID TRAVERSE 1800#3	1	1	1	1	1	1
O, N, EOB, LABEL IGNORED 3404#0	1	1	1	1	1	1
N, LABEL EXCLUDE FROM IGNORANCE 3451#4	0	0	0	0	0	0
G71/G72 SPEED-UP OF RETURNING 5108#1	1	1	1	1	1	1
CANNED CYCLE FOR DRILLING SPEED-UP 5160#0	1	1	1	1	1	1
GOTO/LGOTO SPEED-UP(FIXED TYPE) 6000#1	1	1	1	1	1	1
GOTO/LGOTO SPEED-UP(HISTORY TYPE) 6000#4	1	1	1	1	1	1
FEED FORWARD FOR SKIP FUNCTION 6207#3	1	1	1	1	1	1
FEED FORWARD FOR PMC AXIS CONTROL 8004#3	1	1	1	1	1	1
FEED FORWARD FOR PMC AXIS CONTROL G0 8004#4	1	1	1	1	1	1
G0 BLOCK OVERLAP WITHOUT MOVEMENT 11602#1	1	1	1	1	1	1
ACC/DEC BEFORE INTERPOLATION OF RAPID 19501#5	0	0	0	0	0	0

Table F.2 (b) Parameters displayed on the FCS screen "Axis"

Item Name / Parameter number	0*i*-MF Plus BT30 Standard value	BT40 Standard value	BT50 Standard value	0*i*-TF Plus Auto Lathe Standard value	8” Chuck Standard value	12” Chuck Standard value
T2 USED FOR BELL-SHAPED ACC/DEC G0 [msec] 1621	*1	*1	*1	*1	*1	*1
DEC. RATIO IN RAPID TRAVERSE OVERLAP [%] 1722	30	30	30	30	30	30
DEC. RATIO IN THREADING CYCLE OVERLAP [%] 1726	10	10	10	10	10	10

Item Name / Parameter number	0*i*-MF Plus BT30 Standard value	0*i*-MF Plus BT40 Standard value	0*i*-MF Plus BT50 Standard value	0*i*-TF Plus Auto Lathe Standard value	0*i*-TF Plus 8" Chuck Standard value	0*i*-TF Plus 12" Chuck Standard value
SERVO LOOP GAIN [0.01/sec] 1825	*2	*2	*2	*2	*2	*2
IN-POSITION WIDTH [mm] 1826	0.100	0.100	0.100	0.100	0.100	0.100
VELOCITY CONTROL METHOD 2003#3	1	1	1	1	1	1
FEED-FORWARD FUNCTION 2005#1	1	1	1	1	1	1
VELOCITY LOOP HIGH CYCLE MANAGEMENT 2017#7	1	1	1	1	1	1
VELOCITY FEED-FORWARD COEFFICIENT [%] 2069	100	100	100	100	100	100
FEED-FORWARD COEFFICIENT [%] 2092	98.00	98.00	98.00	98.00	98.00	98.00
VELOCITY GAIN OVERRIDE FOR CUTTING [%] 2107	150	150	150	150	150	150
FEED-FORWARD COEFFICIENT FOR CUTTING [%] 2144	100.00	100.00	100.00	0.00	0.00	0.00
VELOCITY FEED-FORWARD COEFFICIENT [%] 2145	100	100	100	100	100	100
POSITION GAIN FOR RAPID TRAVERSE [0.01/sec] 2178	3000	3000	3000	3000	3000	3000
CUTTING/RAPID VELOCITY GAIN SWITCHING 2202#1	1	1	1	1	1	1
CUTTING/RAPID POSITION GAIN SWITCHING 2213#6	1	1	1	1	1	1
CUTTING/RAPID FEED-FORWARD SWITCHING 2214#4	1	1	1	1	1	1
FEED-FORWARD TIMING TUNING 2415#1	1	1	1	1	1	1
ACCEPTABLE INNER CIRCLE G0-G0 [mm] 10740	1.000	1.000	1.000	1.000	1.000	1.000
ACCEPTABLE INNER CIRCLE G0-G1 [mm] 10741	0.100	0.100	0.100	0.100	0.100	0.100
ACCEPTABLE INNER CIRCLE G1-G0 [mm] 10742	0.100	0.100	0.100	0.100	0.100	0.100
SMART OVERLAP 11236#1	1	1	1	1	1	1

Item Name / Parameter number	0*i*-MF Plus BT30 Standard value	0*i*-MF Plus BT40 Standard value	0*i*-MF Plus BT50 Standard value	0*i*-TF Plus Auto Lathe Standard value	0*i*-TF Plus 8” Chuck Standard value	0*i*-TF Plus 12” Chuck Standard value
SMART OVERLAP TIME CONSTANT [msec] / 11248	16	16	16	16	16	16

Table F.2 (c) Parameters displayed on the FCS screen "Spindle"

Item Name / Parameter number	0*i*-MF Plus BT30 Standard value	0*i*-MF Plus BT40 Standard value	0*i*-MF Plus BT50 Standard value	0*i*-TF Plus Auto Lathe Standard value	0*i*-TF Plus 8” Chuck Standard value	0*i*-TF Plus 12” Chuck Standard value
SHORTCUT FUNCTION FOR ORIENTATION / 4017#7	1	1	1	1	1	1
ORIENTATION TYPE / 4018#3	1	1	1	1	1	1
OPTIMUM ORI: VELOCITY FEED-FORWARD / 4018#4	1	1	1	1	1	1
OPTIMUM ORI: VELOCITY FEED-FORWARD / 4018#5	1	1	1	1	1	1
ORIENTATION TYPE / 4018#6	0	0	0	0	0	0
POSITION GAIN ON ORIENTATION (HIGH) [0.01/sec] / 4060	3000	3000	3000	3000	3000	2000
POS-GAIN ON ORIENTATION (MEDIUM HIGH) [0.01/sec] / 4061	3000	3000	3000	3000	3000	2000
POS-GAIN ON ORIENTATION (MEDIUM LOW) [0.01/sec] / 4062	3000	3000	3000	3000	3000	2000
POSITION GAIN ON ORIENTATION (LOW) [0.01/sec] / 4063	3000	3000	3000	3000	3000	2000
FEED-FORWARD COEFFICIENT [%] / 4344	98.00	98.00	98.00	98.00	98.00	98.00
MAGNETIC FLUX BOOST FUNCTION / 4353#6	1	1	1	1	1	1
CUTTING/RAPID FEED-FORWARD FUNCTION / 4542#4	1	1	1	1	1	1
FEED-FORWARD COEFFICIENT FOR CUTTING [%] / 4627	100.00	100.00	100.00	100.00	100.00	100.00

*1 : The standard value of "T2 USED FOR BELL-SHAPED ACC/DEC G0" (parameter No. 1621) is calculated from the setting of acceleration (parameter No. 1420, No. 1620) and the standard value of rapid traverse jerk.

The standard value of rapid traverse jerk [m/sec^3]

0i-MF Plus	BT30	BT40	BT50
	250	60	20
0i-TF Plus	Auto Lathe	8”Chuck	12” Chuck
	650	100	70

Formula : The standard value of "T2 USED FOR BELL-SHAPED ACC/DEC G0" (parameter No. 1621) [msec] = (No.1420/60/1000)[mm/msec] / No.1620[msec] / The standard value of rapid traverse jerk [mm/msec^3]
When a very large value is set for acceleration, the time constant T2 used for bell-shaped acceleration/deceleration in rapid traverse may increase. In that case, check the setting of acceleration together.

*2 : For a semi-closed system (The bit 1 (OPTx) of parameter No. 1815 is 0), the standard value is 5000.
For a full-closed system (The bit 1 (OPTx) of parameter No. 1815 is 1), the standard value is 3000.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11525	FAXx							

[Input type] Parameter input
[Data type] Bit axis

#7 FAXx In Fast cycle-time setting, the batch setting (Axis) of recommended values is :
0: not executed.
1: executed.

	#7	#6	#5	#4	#3	#2	#1	#0
11527	FSPs							

[Input type] Parameter input
[Data type] Bit spindle

#7 FSPs In Fast cycle-time setting, the batch setting (Spindle) of recommended values is :
0: not executed.
1: executed.

	#7	#6	#5	#4	#3	#2	#1	#0
24745								FCS

[Input type] Parameter input
[Data type] Bit

#0 FCS The adjustment by Fast cycle-time setting is :
0: not completed.
1: completed.

> **NOTE**
> When this parameter is 0, Fast cycle-time setting screen (FCS screen) is displayed. Moreover, "ADJUST" is blinked in the status section. Set this parameter to 1 before shipment of machines and hide FCS screen and "ADJUST" of status section.

F.3 FINE SURFACE SETTING

Overview

FANUC’s recommended values can be set by batch in parameters related to high-quality machining. Three kinds of parameter settings (called setting patterns) are available for several parameters (called selectable parameters). Moreover, operators can select an effective setting pattern from machine tool builders’ made patterns depending on the machining conditions. For these reasons, this function makes it easier to set parameters related to high-quality machining and to choose parameters according to the machining conditions.

Two screens are available in Fine surface setting; Fine surface setting screen, and Pattern select screen. Setting patterns are prepared in Fine surface setting screen. A setting pattern is selected in Pattern select screen.

This manual describes the preparation for setting patterns.

> **NOTE**
> This function is available for only machining center system.

Explanation

Setting patterns are prepared with the following steps 1 to 7.

Step 1
Display of Fine surface setting screen

↓

Step 2
Selection of machine size

↓

Step 3
Batch setting of recommended values

↓

Step 4
Preparation of setting patterns

↓

Step 5
Easy setting of parameters

↓

Step 6
Setting of pattern selection

↓

Step 7
Setting of tolerance

Step 1. Display of Fine surface setting screen

Fine surface setting is enabled to set bit 7(FSD) of parameter No.25000 to 1. And, Fine surface setting screen is displayed when bit 7(FSS) of parameter No.24745 is set to 0.

NOTE

1. The following functions cannot be used simultaneously with Fine surface setting. If Fine surface setting is enabled, the following functions are disabled.
   - Machining condition selection function
   - Machine setting switching function
2. Setting screen for reducing cycle time and Fine surface setting can be used together. However, the following items of setting screen for reducing cycle time cannot be used if both of the two functions are enabled.
   - Speedup of drill cycle, Smart overlap, and Automatic setting of overlap time on "CUTTING BLOCK ADJUSTMENT" screen
   - AI contour control and Smooth tolerance control on "CYCLE ACCURACY ADJUSTMENT" screen
3. After enabling Fine surface setting, basic parameters are enabled before selecting setting pattern, because any of patterns 1 to 3 has not been selected at this time. For the details of basic parameters, refer to the following item of "Parameters set in Fine surface setting".
4. When bit 7(FSS) of parameter No.24745 is set to 0, Adjustment of Fine surface setting is assumed to be not finished, and "ADJUST" blinks in the status section. Set bit 7(FSS) of parameter No.24745 to 1 definitely before a machine shipment, and hide Fine surface setting screen.

Display of Fine surface setting screen and pattern select screen

How to display Fine surface setting screen

1 Press function key SYSTEM several times、Fine surface setting screen is displayed.

Fig. F.3 (a) Fine surface setting screen

2 Press soft key [OPRT].

3 Press soft key [PATTERN NAME], Fine surface setting screen (pattern name setting) is displayed.

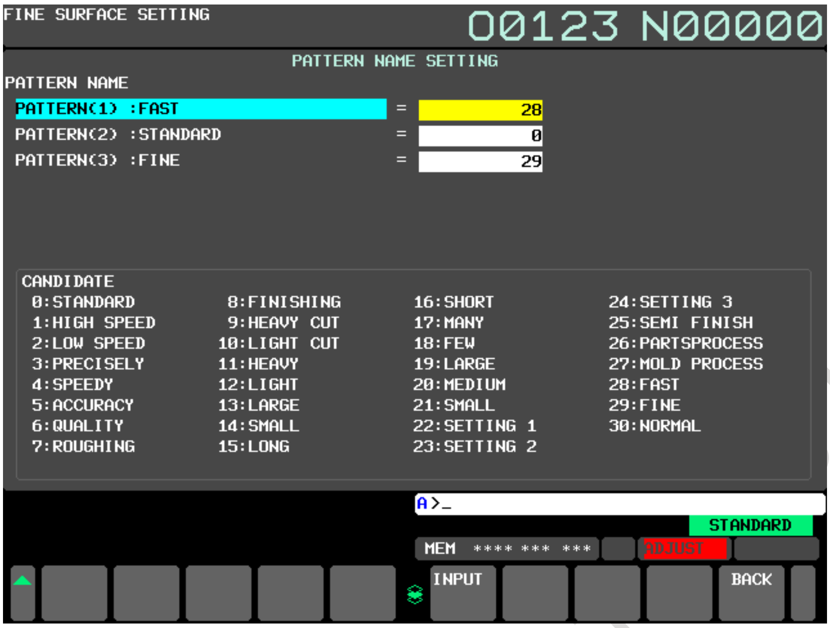


Fig. F.3 (b) Fine surface setting screen (pattern name setting)

How to display pattern select screen

1 Press function key several times、 pattern select screen is displayed.

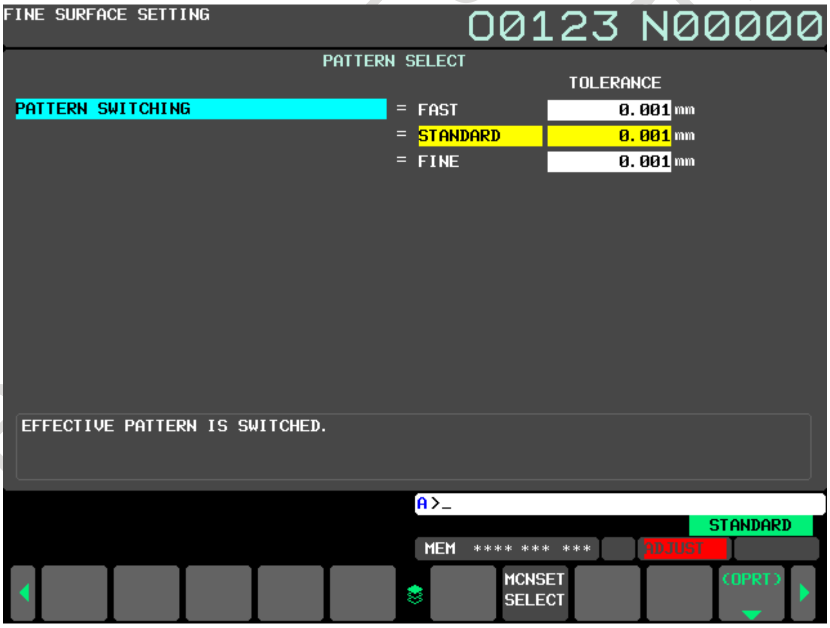


Fig. F.3 (c) Pattern select screen

Step 2. Selection of machine size

> **NOTE**
> The machine size is shared with Fast cycle-time setting. When the machine size is changed in Fine surface setting screen, the machine size in Fast cycle-time setting screen is changed.

1 Press soft key[CHANGE M.SIZE] in Fine surface setting screen, and select machine size. The selecting machine size is displayed on Fine surface setting screen.

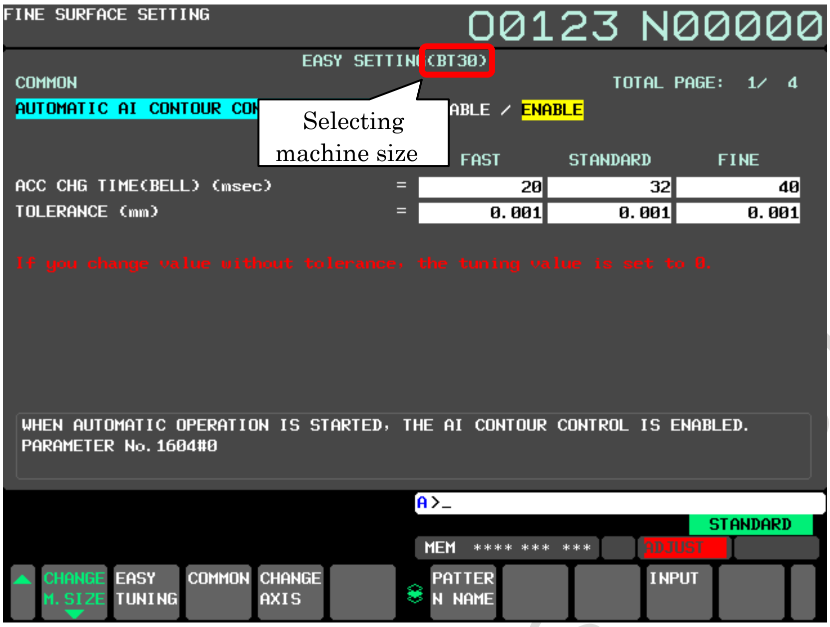


Fig. F.3 (d) Selecting machine size (Fine surface setting screen)

Step 3. Batch setting of recommended values

FANUC's recommended values can be set by batch in parameters related to high-quality machining. Nine patterns of recommended values are available; three patterns of FAST, NORMAL, and FINE for each of spindle taper size BT30, BT40 and BT50.

NOTE

1. Complete basic settings such as axis configuration on parameter screen or parameter adjustment screen before batch setting. The recommended values are not input if the basic settings are not completed.
2. If the following axis is included with the target axes of batch setting, an alarm, DS2075, "ILLEGAL AXIS FOR BATCH SETTING" occurs and batch setting is not started.
   - Incremental system is IS-E
   - Parameter IESPx (Bit 7 of No.1013) is set to 0

   Remove the axis from the target axes and retry batch setting in this case.
3. An alarm, DS2076, "BATCH SETTING CANNOT BE DONE" may be issued with batch setting. In this case, retry the batch setting.
4. Setting parameter MBEx (Bit 0 of No.25002) to 1 starts batch setting regardless of parameter PWE (Bit 0 of No.8900) and 8-Level data protection function.
5. Parameter SYPx (Bit 4 of No.8303) enables automatic parameter setting of the slave axis in Axis synchronous control. If this parameter is set to 1, the recommended value for the master axis is set for the slave axis as for "Time constant of acc./dec. after cutting feed interpolation in acc./dec. before interpolation mode" in Fine surface setting, even though the slave axis is not the target axis of batch setting.
6. Batch setting overwrites current settings with the recommended values. Don't apply the batch setting if the parameters related to high-quality machining have already been set.
7. After batch setting, 28 "FAST", 0 "STANDARD", 29 "FINE" are effective for pattern (1), (2), (3) on Fine surface setting (pattern name setting) screen.
8. After batch setting, pattern 2 is selected as an effective setting pattern.

1 Select target axes of batch setting by parameter MBE (Bit 0 of No.25002). The axes with this parameter set to 1 are the target axes. An alarm, PW0000, "POWER MUST BE OFF" occurs when parameter MBE is set for at least one axis. Turn the power off and on again after setting all target axes.

2 When turning the power on again with MBE set to 1 in at least one axis, the batch setting is started in power-on of the CNC. For the target parameters of the batch setting, refer to the item "Parameters set in Fine surface setting". For axis type parameters, the recommended values are set only in the axis with parameter MBE set to 1. For path type parameters, the recommended values are set only in the path including the axis with MBE set to 1. For the parameters common to the system, the recommended values are set when the batch setting is completed.

3 Parameter MBE is turned to 0 when turning the power on again, regardless of whether or not the batch setting succeeded.

Example)

In the system with the following axis configuration, recommended values are set as follows when setting 1 to parameter MBE in X2, Z2, Y3 axes.

Table F.3 (a) Example of batch setting

	Parameter MBEx	Recommended values		
		Common to the system	Path type	Axis type
X1	0			Not set
Y1	0		Not set	Not set
Z1	0			Not set
X2	1			**Set**
Y2	0	**Set**	**Set**	Not set
Z2	1			**Set**
X3	0			Not set
Y3	1		**Set**	**Set**
Z3	0			Not set

Step 4. Preparation of setting patterns

Select pattern name according to machining condition, and adjust parameters of each patterns.

1 Select the names of pattern 1 to 3 in Fine surface setting (pattern name setting) screen. Refer to the item of “Setting pattern name” for the details.
2 Input setting data in selectable parameters in Fine surface setting screen. Refer to the item of “Parameters set in Fine surface setting” for selectable parameters.

NOTE
1. Setting data cannot be input on Fine surface setting screen if parameter PWE is set to 0. A warning “WRITE PROTECT” is issued along with setting data input.
2. Setting data cannot be input to the item of parameter input on Fine surface setting screen if parameters are protected by 8-Level data protection function. A warning “WRITE PROTECT” is issued along with the setting data input.
3. Setting data cannot be input to the item of setting input on Fine surface setting screen if setting parameters are protected by 8-Level data protection function. A warning “WRITE PROTECT” is issued along with setting data input.
4. Setting data must be input on Fine surface setting screen during emergency stop or in MDI mode with automatic operation stopped. A warning “CNC RUNNING” is issued along with setting data input during automatic operation, and a warning “SWITCH TO MDI MODE” is issued along with setting data input in other modes.
5. Parameter SYPx (Bit 4 of No.8303) enables automatic parameter setting for the slave axis in Axis synchronous control. If this parameter is set to 1, the recommended data for the master axis is set for the slave axis as for “Time constant of acc./dec. after cutting feed interpolation in acc./dec. before interpolation mode” in Fine surface setting.
6. Fine surface setting screen is not displayed in a loader path if Loader control function is enabled.

Operation in Fine surface setting screen

Select name of setting pattern

1 Display Fine surface setting screen(Pattern name setting).
2 Press cursor key ⬆ ⬇ to move cursor to the setting pattern to be selected.
3 Input a setting to key input buffer, and press input key INPUT or soft key [INPUT].

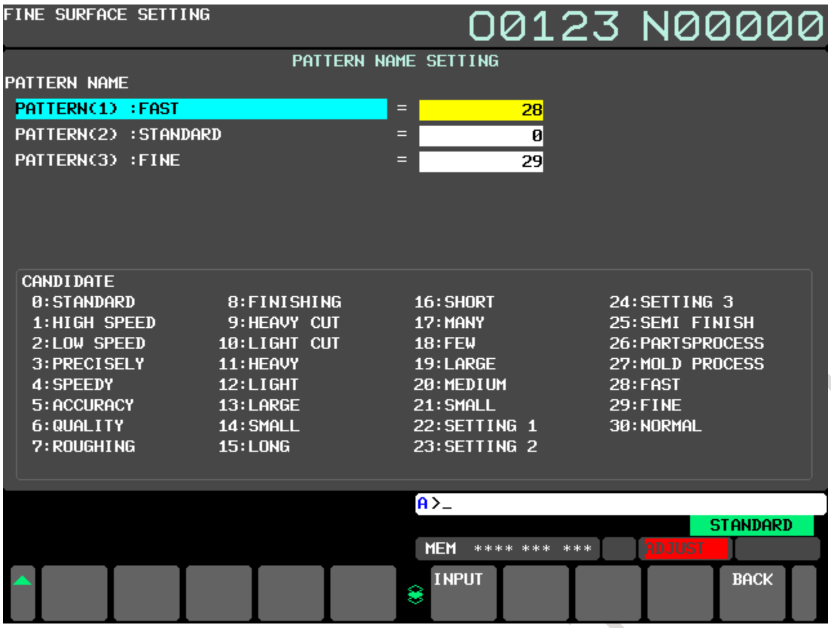


Fig. F.3 (e) Fine surface setting screen (pattern name setting)

Input parameters of each setting patterns

1 Display Fine surface setting screen.
Setting items which common to the path are displayed when press soft key[COMMON], setting items of each axes are displayed when press soft key[CHANGE AXIS]. Displayed axis is switched when press soft key[CHANGE AXIS] several times. Press page change keys to move to the screen of the axis to be set.

2 Press cursor keys to move the cursor to the parameter to be set.

3 Input a setting data to key input buffer, and press input key or soft key [INPUT]. Pressing soft key [ALL AXES] input common setting data by batch to all axes in the path.

Fig. F.3 (f) Fine surface setting screen (common)

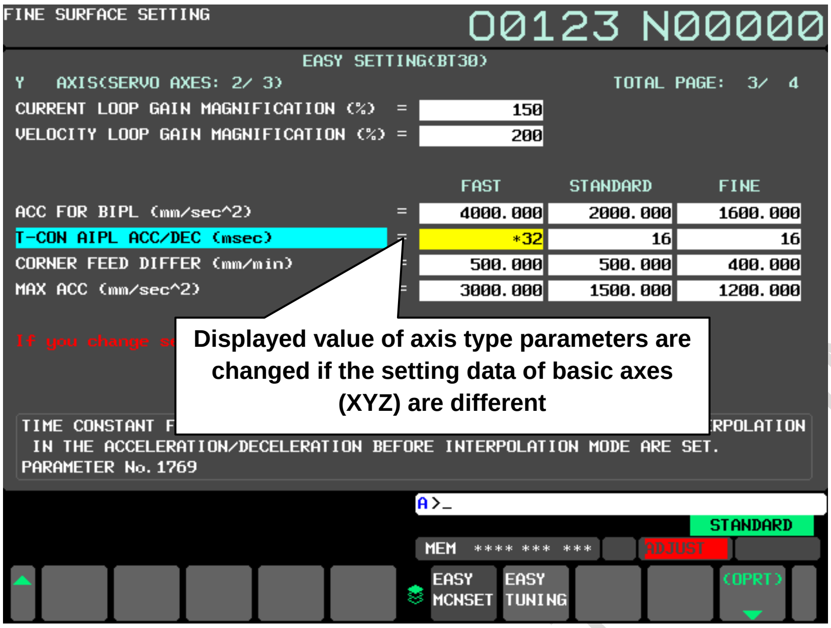


Fig. F.3 (g) Fine surface setting screen (each axes)

Step 5. Easy tuning of parameters

Tuning which set selectable parameters more high-precision/high-speed can be done in easy tuning screen.

1 Prepare setting pattern by finishing steps 1 to 4.
2 Set MAEx(bit 0 of parameter No.25003) to 0 for target axes of easy tuning.
3 Tune parameters in easy tuning screen. About the tuning parameters, refer to the item “Tuning parameters”

> **NOTE**
> 1. Easy tuning cannot be done when parameter PWE is set to 0. A warning “WRITE PROTECT” is issued along with tuning in this state.
> 2. Easy tuning cannot be done when parameters are protected by 8-Level data protection function. A warning “WRITE PROTECT” is issued along with tuning in this state.
> 3. Easy tuning must be done during emergency stop or in MDI mode with automatic operation stopped. A warning “CNC RUNNING” is issued along with tuning during automatic operation and a warning “SWITCH TO MDI MODE” is issued along with tuning input in other modes.
> 4. Easy tuning screen is not displayed when parameter MAE of all axes in a path is set to 1.

Display easy tuning screen

How to display easy tuning screen

1 Display Fine surface setting screen, and press soft key[EASY TUNING].

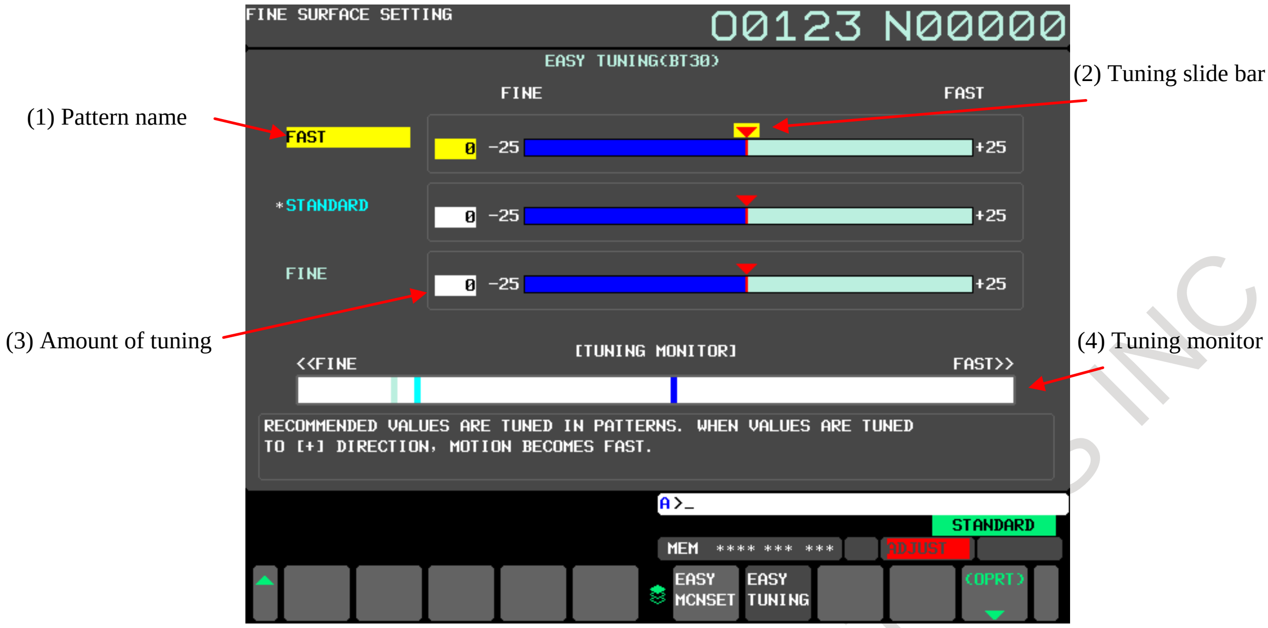


Fig. F.3 (h) Easy tuning screen

Explanation of screen

(1) Pattern name	:	Name of pattern 1 to 3 are displayed. It is displayed in order of pattern 1, pattern 2, and pattern 3.
(2) Tuning slide bar	:	Present amount of tuning is shown by the arrow. When the setting is more high-speed, the arrow moves to left. And, when the setting is more high-precision, the arrow moves to right.
(3) Amount of tuning	:	Present amount of tuning is shown by the numerical value.
(4) Tuning monitor	:	Relation of each setting pattern is displayed. When the setting is more high-speed, the arrow moves to left. And, when the setting is more high-precision, the arrow moves to right.

Operation in Easy tuning screen

Tuning of setting pattern

1. Press cursor keys ⬆ ⬇ to move the cursor to the setting pattern to be tuned.
2. Press cursor key ⬅. Setting pattern is tuned to high-precision. And, press cursor key ➡. Setting pattern is tuned to high-speed. Moreover, input the amount of tuning to key input buffer, and press input key INPUT or soft key [INPUT], same tuning can be done.
3. Setting value of setting pattern after tuning can be confirmed in Fine surface setting screen.

NOTE

1. A warning “Can’t do more adjustment.” is issued along with tuning as follows and tuning is not done. About the tuning range, refer to the item “Tuning parameters”.
   - Tuning to which setting pattern exceeds the tuning range is done.
   - Tuning is done in the state to which setting pattern has exceeded the tuning range.
2. The position of each setting pattern in tuning monitor is calculated by the setting of tuning parameters of the reference axis. When parameter MAEs of the reference axis is set to 1, the positions of each setting pattern in tuning monitor are fixed on the most high-precision.
3. When the setting value of tuning parameters is changed after doing easy tuning, amount of tuning of setting pattern which is changed becomes 0. When easy tuning is done again in this state, tuning is done by the setting of tuning parameters after changing.
4. When the setting value of parameter MAE is changed after doing easy tuning, amount of tuning of all setting patterns become 0. When easy tuning is done again in this state, tuning is done by the setting of tuning parameters after changing.

Step 6. Setting of setting pattern selection

The effective setting pattern can be selected by screen operation, program commands, and signal operation.

Pattern select screen is not displayed when parameter MPS (Bit 3 of No.25001) is set to 1. To protect setting patterns from being selected by operators, set parameter MPS equal to 1.

NOTE

If Peripheral axis control is valid, the setting pattern of Peripheral axis control follows the setting pattern of the path specified with parameter No.3040 to 3042. The setting pattern of Peripheral axis control is preserved as it is started. Even if the setting pattern specified with parameter No.3040 to 3042 is changed after Peripheral axis control is started, the actual setting pattern is not changed.

Step 6-1. Selection by screen operation

Selection of setting pattern by screen operation

The effective setting pattern can be selected on pattern select screen. Setting parameter MPD (Bit 1 of No.25001) equal to 1 enables to check the setting data of each setting pattern on this screen. The way to switch between common and each axes is same as Fine surface setting screen.

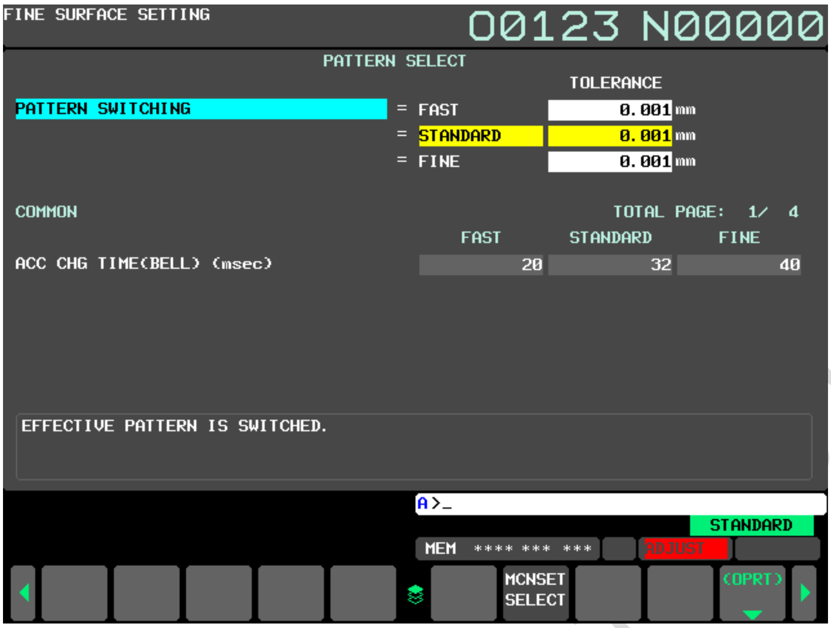


Fig. F.3 (i) Display setting values of setting pattern (common)(pattern select screen)

Fig. F.3 (j) Display setting values of setting pattern (each axes)(pattern select screen)

NOTE

1. Pattern selection by pattern select screen must be performed during emergency stop or in MDI mode with automatic operation stopped. A warning “CNC RUNNING” is issued along with the pattern selection during automatic operation, and a warning “SWITCH TO MDI MODE” is issued along with the pattern selection in other modes.
2. Pattern select screen is not displayed in a loader path if loader control function is valid.

Operation in pattern select screen

Selection of setting pattern

1 Press cursor keys ← ↑ ↓ → to move cursor to the setting pattern to be enabled.

2 Press soft key [SELECT], and the selected setting pattern is turned to valid.

Step 6-2. Selection by program command

Selection of setting pattern by G08.1 command

The effective setting pattern can be selected by commanding G08.1 during automatic operation. The setting pattern to be enabled is specified with address P. Refer to the following item of “Format” for the format of G08.1 command.

Selection of setting pattern by G05.1, G05 and G08.1 command

A setting pattern can be selected by the G code of Machining condition selection function (G05.1, G05, G08 commands) by setting parameters No.25008 and No.25009 to specify the range of accuracy level in each of pattern 1 to 3. This enables to use the machining programs without any changes, which has been used on the machines with Machining condition selection function available. Refer to the following item of “Format” for the format of G05.1, G05, G08 commands.

The range of pattern 1 and pattern 3 are specified with parameter No.25008 and No.25009, respectively.
Example)
To specify accuracy level 1 to 3 with pattern 1, level 4 to 7 with pattern 2, and level 8 to 10 with pattern 3, set 3 in parameter No.25008 and 8 in No.25009.

Setting pattern	Pattern 1			Pattern 2				Pattern 3		
Accuracy level	1	2	3	4	5	6	7	8	9	10

No.25008 No.25009

Step 6-3. Selection by signal operation

Selection of setting pattern by signal operation

The effective setting pattern can be selected by signal input.

< Setting pattern select signal MSSP1, MSSP2<Gn589.5, 6> >
A setting pattern is specified with MSSP1, MSSP2.
Set MSSP1, MSSP2 as follows.

Table F.3 (b) Setting pattern select signal

Setting pattern	Gn589	
	bit 6	bit 5
Pattern 1	0	1
Pattern 2	1	0
Pattern 3	1	1

< Setting pattern switch signal MSWSP<Gn589.7> >
Setting MSWSP equal to “1” enables the setting pattern specified by MSSP1, MSSP2.
< Setting pattern switch complete signal MSWFN<Fn537.6> >
After setting MSWSP to “1” and switching setting pattern, MSWFN is turned to “1”. When setting MSWSP to “0”, MSWFN is turned to “0”.

NOTE

1. In program operation, switch the effective setting pattern while waiting for FIN signal from mask buffering M codes. Note that these M codes must be commanded singly. An alarm, PS2083, "ILLEGAL PATTERN SELECT" occurs if MSWSP is set to "1" at other timing.
2. If MSWSP is set to "1" while either axis in the path is moving, an alarm, PS2083, "ILLEGAL PATTERN SELECT" is issued.
3. When both of Fine surface setting and Machine setting switching function are disabled, setting MSWSP to "1" issues an alarm, PS2089, "MACHINE SET SWITCHING INVALID".
4. Setting patterns does not switch when both MSSP1 and MSSP2 are set to "0", but MSWFN is turned to "1".

Output effective setting pattern

The current setting pattern can be output with signals.

< Setting pattern output signal MSPO1, MSPO2<Fn537.4, 5> >

The current effective setting pattern is output to MSPO1, MSPO2. Refer to the following table for the detail of output signals.

Table F.3 (c) Setting pattern output signal

Pattern	Fn537	
	bit 5	bit 4
Basic parameter or Fine surface setting is disabled	0	0
Pattern 1	0	1
Pattern 2	1	0
Pattern 3	1	1

Step 6-4. Selection by other way

Selection of setting pattern by power-off and reset

If a value between 1 and 3 is set in parameter No.25010, the setting pattern specified with the parameter is selected as an effective setting pattern in power-off and reset operation. The value between 1 and 3 set to parameter No.25010 corresponds to each pattern 1 to 3. If a value other than 1 to 3 is set to parameter No.25010, previously effective setting pattern is preserved in power-off or reset.

Selection way and display effective setting pattern

Relation between selection way and effective pattern

The following figure shows the relation between each selection way of setting pattern and the effective parameters.

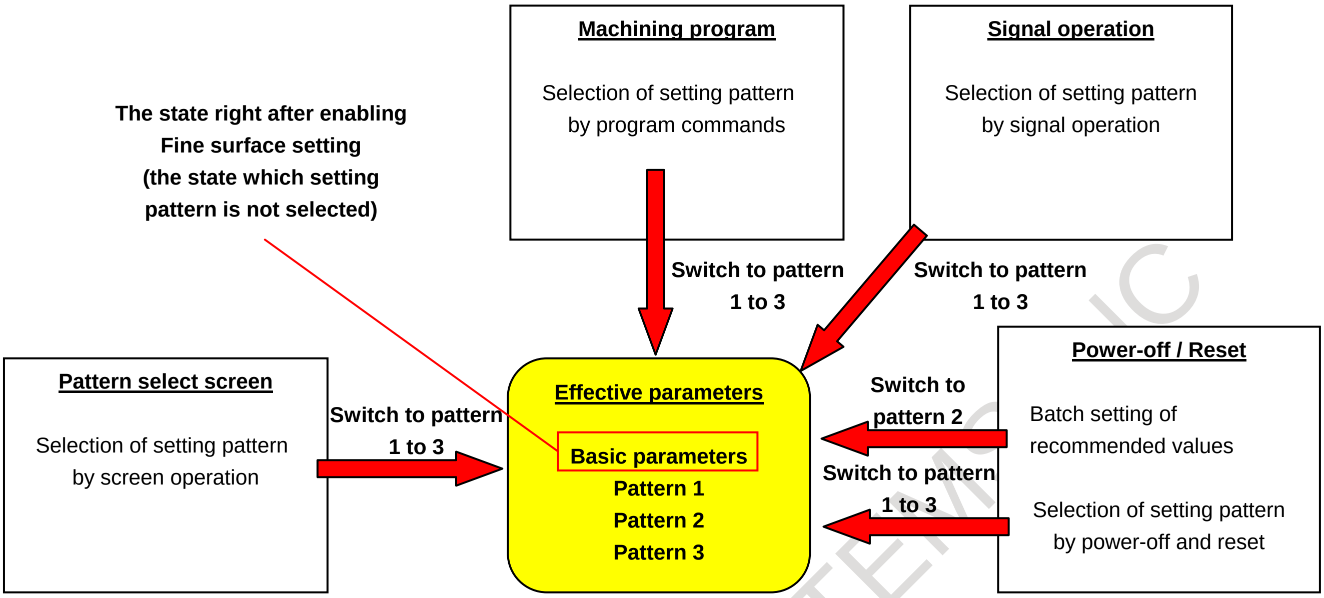


Fig. F.3 (k) Relation between selection way and effective patterns

Display effective setting pattern

The effective setting pattern is displayed at warning message area. The effective setting pattern displayed at the warning message area is hidden if parameter MPW (Bit 4 of No.25001) is set to 1. Set parameter MPW to 1 so that operators cannot see the setting pattern.

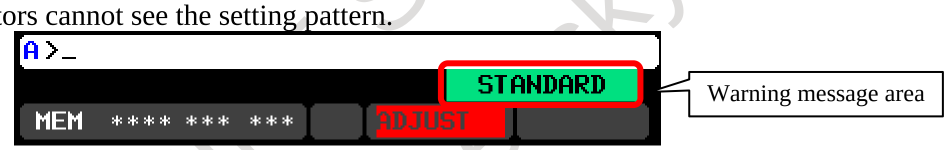


Fig. F.3 (l) Display effective setting pattern

Step 7. Setting of tolerance

The tolerance value in Smooth tolerance control+ can be set in pattern select screen.

> **NOTE**
> This setting is unnecessary if Smooth tolerance+ control is not used.

Setting of Tolerance

The tolerance value of Smooth tolerance control+ (parameter No.11786) can be set to each setting patterns in pattern select screen.

> **NOTE**
> 1. The item of tolerance is not displayed if Smooth tolerance+ control is disabled.
> 2. Setting data cannot be input on pattern select screen if setting parameters are protected by 8-Level data protection function. A warning "WRITE PROTECT" is issued along with setting data input.
> 3. Setting data must be input on pattern select screen during emergency stop or in MDI mode with automatic operation stopped. A warning "CNC RUNNING" is issued along with setting data input during automatic operation, and a warning "SWITCH TO MDI MODE" is issued along with setting data input in other modes.
> 4. Pattern select screen is not displayed in a loader path if loader control function is valid.

Operation in pattern select screen

Input setting data to tolerance

1 Press cursor keys [←] [↑] [↓] [→] to move cursors to the tolerance to be set.

2 Input setting data to key input buffer, and press input key [INPUT] or soft key [INPUT].

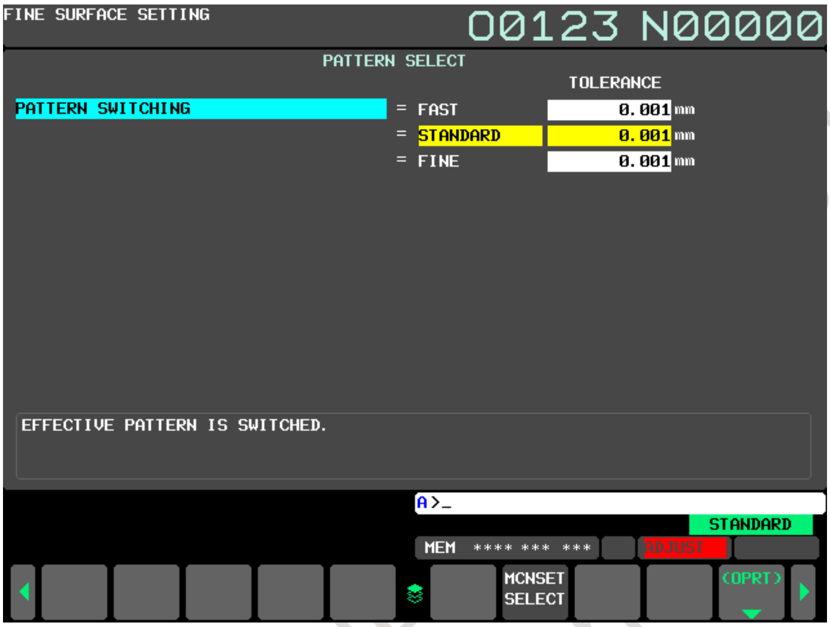


Fig. F.3 (m) Setting of tolerance (pattern select screen)

Appendix

Parameters set in Fine surface setting

The following table shows the parameters set in Fine surface setting related to high-quality machining.

Table F.3 (d) Parameters set in Fine surface setting

Items	Basic parameters	Pattern 1	Pattern 2	Pattern 3	Notes
Unit of recommended value	Recommended value BT30/40/50	Recommended value BT30/40/50	Recommended value BT30/40/50	Recommended value BT30/40/50	
AI contour control with automatic operation start	No.1604#0	-	-	-	(*1)
-	1	-	-	-	
Current loop gain magnification for cutting	No.2334	-	-	-	(*1)
[%]	150	-	-	-	
Velocity loop gain magnification for cutting	No.2335	-	-	-	(*1)
[%]	200	-	-	-	
Max allowable rate in acc./dec. before interpolation	No.1660	-	-	-	(*2)
[mm/sec^2]	2000 / 800 / 620	4000 / 2800/ 1580	2000 / 800 / 620	1600 / 400 / 300	
Acceleration change time of bell-shaped acc./dec. before interpolation	No.1772	-	-	-	(*2)
[msec]	32 / 32 / 60	20 / 20 / 48	32 / 32 / 60	40 / 40 / 64	

Items	Basic parameters	Pattern 1	Pattern 2	Pattern 3	Notes
Unit of recommended value	**Recommended value BT30/40/50**	**Recommended value BT30/40/50**	**Recommended value BT30/40/50**	**Recommended value BT30/40/50**	
Acc./dec. type after interpolation in acc./dec. before interpolation mode	No.1602#3 No.1602#6	-	-	-	(*1)
-	1, 1	-	-	-	
Time constant for acc./dec. after cutting feed interpolation in acc./dec. before interpolation mode	No.1769	-	-	-	(*2)
[msec]	16 / 16 / 24	24 / 24 / 24	16 / 16 / 24	16 / 16 / 24	
Time constant for acc./dec. after cutting feed interpolation in acc./dec. before interpolation mode	No.1783	-	-	-	(*2)
[mm/min]	500 / 300 / 248	500 / 500 / 632	500 / 300 / 248	400 / 150 / 120	
Max allowable acc. rate for deceleration function based on acc. rate	No.1735 No.1737	-	-	-	(*2)
[mm/sec^2]	1500 / 600 / 465	3000 / 2100/ 1185	1500 / 600 / 465	1200 / 300 / 225	
Min allowable feedrate for deceleration function based on acc. rate	No.1732 No.1738	-	-	-	(*1)
[mm/min]	100	-	-	-	
Smooth speed control	No.19503#0	-	-	-	(*1)
-	1	-	-	-	(*A)
Invalidation of speed control with change of acc. rate by block length in Smooth speed control	No.19517#0	-	-	-	(*1) (*A)
-	1	-	-	-	
Invalidation of speed control with acc. rate and change of acc. rate	No.19517#1	-	-	-	(*1)
-	1	-	-	-	

Items (*1) are the target of batch setting of recommended values.
Items (*2) are the target of batch setting of recommended values and selectable parameters.
Items (*A) are the target of batch setting of recommended values if AI contour control II is enabled.

NOTE

1. For items (*1) and (*2), the following parameters are called basic parameters.
   (*1): Parameters effective regardless of setting pattern selection
   (*2): Parameters effective with no setting pattern selected
2. The recommended values are varied depending on the unit of parameters and the minimum command/machine unit.
   Ex.)The minimum allowable feedrate for deceleration function by acceleration rate is a parameter with machine unit.
   The recommended value is 100 [mm/min] for millimeter machines.
   It is 3.937 [inch/min] for inch machines, on the other hand.
3. After batch setting of recommended values, items (*1) and (*2) are set as follows.
   (*1): The recommended values in the table above are set to the parameters in column "Basic parameter".
   (*2): The recommended values in the table above are set to the parameters in column "Basic parameter" and "Pattern 1 to 3".
4. Items(A*) are not displayed on Fine surface setting screen and pattern select screen if AI contour control II is disabled.

Tuning parameters

In parameters related to high-quality machining, the parameters (tuning target) which are tuned by easy tuning, the parameters (calculation target) which are used for calculation of tuning monitor, and the tuning range of each parameter is as follows.

Table F.3 (e) Tuning parameter

Item	Basic parameter	Tuning target	Calc target	Tuning range
Max allowable rate in acc./dec. before interpolation	No.1660	O	O	0.0<n≦5000.0 [mm/s^2] *
Acceleration change time of bell-shaped acc./dec. before interpolation	No.1772	O	O	7<n≦64 [ms]
Time constant for acc./dec. after cutting feed interpolation in acc./dec. before interpolation mode	No.1769	-	O	-
Max allowable feedrate difference for feedrate determination based on corner feedrate difference	No.1783	O	O	0.0<n≦1000.0 [mm/min] *
Max allowable acc. rate for deceleration function based on acc. rate	No.1735 No.1737	O	O	0.0<n≦3750.0 [mm/s^2] *

NOTE

When increment system is IS-E, tune the * parameters within the range of 0.0<n≤999.0.

Setting pattern name

The following table shows the relation between the setting data in Fine surface setting (pattern name setting) screen and the name of setting pattern.

Table F.3 (f) Name of setting pattern

Setting data in Fine surface Setting (pattern name setting) screen	Name of setting pattern
0	STANDARD
1	HIGH SPEED
2	LOW SPEED
3	PRECISELY
4	SPEEDY
5	ACCURACY
6	QUALITY
7	ROUGHING
8	FINISHING
9	HEAVY CUT
10	LIGHT CUT
11	HEAVY
12	LIGHT
13	LARGE
14	SMALL
15	LONG
16	SHORT
17	MANY
18	FEW
19	LARGE
20	MEDIUM
21	SMALL
22	SETTING 1
23	SETTING 2
24	SETTING 3
25	MEDIUMFINING
26	PAERSPROCESS
27	MOLD PROCESS
28	FAST
29	FINE
30	NORMAL

Format

G08.1P_ ;

P_ : Setting pattern
- P1 : “Pattern 1”
- P2 : “Pattern 2”
- P3 : “Pattern 3”

Ex.) When commanding G08.1P3, “Pattern 3” is selected as an effective setting pattern.

NOTE
1. An alarm, PS2083, "ILLEGAL PATTERN SELECT" is issued when processing G08.1 block in the following cases.
   - A value other than 1 to 3 is commanded with address P
   - Either axis in the path is moving
2. G08.1P_ is a G code preventing buffering.
3. If Fine surface setting and Machine setting switching function are disabled, an alarm, PS0010, "IMPROPER G CODE" is issued when commanding G08.1.
4. If Fine surface setting and Loader control function are enabled, an alarm, PS0010, "IMPROPER G CODE" is issued when commanding G08.1 in a loader path.

G05.1 Q1 R_ ;
R_ : level(1 to 10)
G05 P10000 R_ ;
R_ : level(1 to 10)
G08 P1 R_ ;
R_ : level(1 to 10)

NOTE
1. An alarm, PS2083, "ILLEGAL PATTERN SELECT" is issued when processing G05.1Q1, G05P10000, G08P1 blocks in the following cases.
   - A value other than 1 to 3 is commanded with address R
   - Either axis in the path is moving
2. G05.1Q1R_, G05P10000R_, and G08P1R_ are G codes preventing buffering.
3. These commands can be used by setting parameters No.25008, No.25009 to specify the range of pattern 1 to 3. An alarm, PS0010, "IMPROPER G CODE" is issued when commanding G05.1Q1R_, G05P10000R_, or G08P1R_ if parameters No.25008, No.25009 are illegally set.
4. If Fine surface setting and Loader control function are enabled, an alarm, PS0010, "IMPROPER G CODE" is issued when commanding G05.1Q1R_, G05P10000R_, or G08P1R_ in a loader path.

Signal

Setting pattern select signal MSSP1, MSSP2<Gn589.5, 6>

[Classification] Input signal
[Function] Select an effective pattern in setting pattern selection by signal operation
[Operation] Select an effective setting pattern as follows.

Setting pattern	Gn589	
	bit 6	bit 5
Pattern 1	0	1
Pattern 2	1	0
Pattern 3	1	1

NOTE
Setting pattern does not switch when both MSSP1 and MSSP2 are set to “0”, but MSWFN<Fn537.6> is set to “1”.

Setting pattern switch signal MSWSP<Gn589.7>

[Classification] Input signal
[Function] Switch the effective pattern in setting pattern selection by signal operation
[Operation] When setting this signal from “0” to “1”, the combination of spindle axis and servo axis to synchronize (Parameter No.24208) is updated. This signal is valid when Spindle/Servo axis combination change condition signal SPSVSS<F0537.1> is “1”.

Setting pattern output signal MSPO1, MSPO2<Fn537.4, 5>

[Classification] Output signal
[Function] These signals indicate the current effective setting pattern
[Output condition] The output of these signals as follows, depending on the currently effective setting.

Pattern	Fn537	
	bit 5	bit 4
Basic parameters, or Fine surface setting is disabled	0	0
Pattern 1	0	1
Pattern 2	1	0
Pattern 3	1	1

Setting pattern switch complete signal MSWFN<F0537.6>

[Classification] Output signal
[Function] This signal indicates the end of setting pattern switching in setting pattern selection by signal operation.
[Output condition] This signal is turned to “1” in the following case:
- The setting pattern has been switched after setting pattern switch signal MSWSP is set to “1”.

This signal is turned to “0” in the following case:
- Setting pattern switch signal MSWSP is set to “0”.
- The setting pattern has not been switched yet after setting pattern switch signal MSWSP is set to “1”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn589	MSWSP	MSSP2	MSSP1					
Fn537		MSWFN	MSPO2	MSPO1				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
24745	FSS							

[Input type] Parameter input
[Data type] Bit

#7 FSS The adjustment by Fine surface setting is
0: not completed.
1: completed.

> **NOTE**
> When this parameter is 0, Fine surface setting screen (FSS screen) is displayed. Moreover, "ADJUST" is blinked in the status section. Be sure to set this parameter to 1 before shipment of machines and hide FSS screen and "ADJUST" of status section.

	#7	#6	#5	#4	#3	#2	#1	#0
25000	FSD							

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#7 FSD Fine surface setting is
0: enabled.
1: disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
25001				MPW	MPS		MPD	

[Input type] Parameter input
[Data type] Bit

#1 MPD In Fine surface setting, the setting data of each setting pattern is
0: not displayed on pattern select screen.
1: displayed on pattern select screen.

#3 MPS In Fine surface setting, pattern select screen is
0: displayed.
1: not displayed.

#4 MPW In Fine surface setting, the effective setting pattern is
0: displayed on warning message area.
1: not displayed on warning message area.

	#7	#6	#5	#4	#3	#2	#1	#0
25002								MBE

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#0 MBE In Fine surface setting, batch setting of recommended values is
0: disabled.
1: enabled.

> **NOTE**
> Batch setting of recommended values is done when the power is turned on again. Bit 0 (MBE) of parameter No. 25002 is set to 0 after the batch setting is completed.

	#7	#6	#5	#4	#3	#2	#1	#0
25003								MAE

[Input type] Parameter input
[Data type] Bit axis

#0 MAE In Fine surface setting, easy tuning is
0: enabled.
1: disabled.

25008	Accuracy level corresponding to pattern 1 in Fine surface setting
25009	Accuracy level corresponding to pattern 3 in Fine surface setting

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 10

This parameter specifies the accuracy level of machining condition function corresponding to pattern 1 to 3 of Fine surface setting.
Example)
If the precision level 1 to 3, level 4 to 7, and level 8 to 10 correspond to pattern 1, 2, 3, respectively, set parameter No.25008 to 3 and parameter No.25009 to 8.

> **NOTE**
> Set a larger value to parameter No.25009 than to parameter No.25008. If parameter No.25009 is smaller than parameter No.25008, setting pattern cannot be selected in G05.1, G05, G08 commands.

25010	Setting pattern in Fine surface setting in power-off, reset operation

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

This parameter specifies the setting pattern selected in power-off or reset operation during automatic operation. If this parameter is set to 1 to 3, setting pattern 1 to 3 is selected. The previously effective setting pattern is selected, otherwise.

	#7	#6	#5	#4	#3	#2	#1	#0
1013	IESP							

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#7 IESP When the incremental system is IS-C, IS-D, or IS-E, the function to set a larger value to the parameter related to speed and acceleration
0: is not used.
1: is used.

	#7	#6	#5	#4	#3	#2	#1	#0
8303				SYPx				

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#4 SYPx Some parameters must be set to the same value for the master and slave axes in Axis synchronous control. When a value is set in such a parameter for the master axis,
0: the same value is not automatically set for the slave axis.
1: the same value is automatically set for the slave axis.

	#7	#6	#5	#4	#3	#2	#1	#0
8900								PWE

[Input type] Parameter input
[Data type] Bit
#0 PWE The setting, from an external device and MDI panel, of parameters unable to be written by setting input is
0: disabled.
1: enabled.

Alarm and message

番号	Message	Description
PW0000	POWER MUST BE OFF	A parameter has been set for which the power must be OFF then ON again.
PS0010	IMPROPER G-CODE	An unavailable G code is commanded.
PS2083	ILLEGAL PATTERN SELECT	A setting pattern cannot be selected for the following reasons. - A value other than 1 to 3 is specified with address P of G08.1 - A value other than 1 to 10 is specified with address R of G05.1Q1, G05P10000, G08P1 - Either axes in the path is moving - Setting pattern selection by signal operation is commanded during automatic operation, except for the waiting time for FIN of mask buffer M codes

番号	Message	Description
DS2075	ILLEGAL AXIS FOR BATCH SETTING	Batch setting of recommended values cannot be started for the following reason. - The incremental system of the axis is IS-E and parameter IESPx(No.1013#7) is set to 0
DS2076	BATCH SETTING CANNOT BE DONE	Batch setting of recommended values has not been performed. Set parameter MBE (Bit 0 of No.25002) and start the batch setting again.

F.4 OPERATION CHECK

Perform automatic operation and check cycle time and shape of workpiece. Adjust the parameters on each setting screen (FCS screen, FSS screen) as necessary. Parameters set by Fast cycle-time setting and Fine surface setting can be changed directly from each setting screen (FCS screen, FSS screen). For parameters of the axis type and spindle type, the same setting can be made for all axes with one input by using the soft key [ALL SV/SP]. As for the parameters of the axis type and the spindle type, when the parameter values of all axis (basic three axis in the case of the axis type and all spindle in case of the spindle type) are not the same value, an asterisk (*) is displayed.

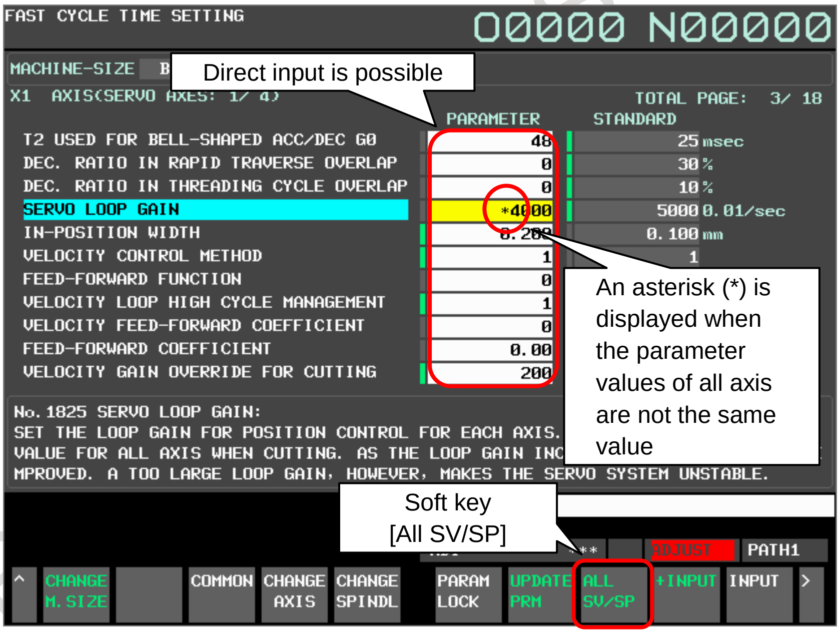


Fig. F.4 (a) Direct input of parameters (FCS screen)

> **NOTE**
> Check operation with all setting pattern of Fine surface setting. For details on how to select the setting pattern, refer to the section "Setting of pattern selection".

⚠ WARNING

By Fast cycle-time setting, the function to reduce the cycle-time, such as the overlap function, becomes valid and the tool path may change. Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

F.5 ADJUSTMENT COMPLETION SETTING

Fast cycle-time setting and Fine surface setting are functions for the machine tool builder. Set adjustment completion before machine shipment and hide FCS screen, FSS screen, and "ADJUST" of status section.

The setting method of adjustment completion is as follows.

- Fast cycle-time setting
  Set the bit 0 (FCS) of parameter No. 24745 to 1 on the parameter screen.
- Fine surface setting
  Set the bit 7 (FSS) of parameter No. 24745 to 1 on the parameter screen.

NOTE

1. Disable Fine surface setting to set FSD(bit 7 of parameter 25000) to 1 when pattern switching of Fine surface setting is not used.
2. Disable Fine surface setting to set FSD(bit 7 of parameter 25000) to 1 when enable Machining condition selection function.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
24745	FSS							FCS

[Input type] Parameter input
[Data type] Bit

#0 FCS The adjustment by Fast cycle-time setting is :
0: not completed.
1: completed.

NOTE

When this parameter is 0, Fast cycle-time setting screen (FCS screen) is displayed. Moreover, "ADJUST" is blinked in the status section. Set this parameter to 1 before shipment of machines and hide FCS screen and "ADJUST" of status section.

#7 FSS The adjustment by Fine surface setting is :
0: not completed.
1: completed.

> **NOTE**
> When this parameter is 0, Fine surface setting screen (FSS screen) is displayed. Moreover, "ADJUST" is blinked in the status section. Set this parameter to 1 before shipment of machines and hide FSS screen and "ADJUST" of status section.

INDEX

<Number>

<A>

<B>

<C>

<D>

<E>

<P>

<Q>

<R>

<S>

<T>

<U>

<V>

<W>

<Y>

REVISION RECORD

Edition	Date	Contents
01	Feb., 2019	

B-64693EN-1/01

* B - 6 4 6 9 3 E N - 1 / 0 1 *